2026R1

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

HFSS™ 3D Layout 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 HFSS 3D Layout Help **1-1**

Workflow of an HFSS 3D Layout Design 1-1

Recording a Script to a File 1-3

Inserting an HFSS 3D Layout Design 1-3

Inserting an EM Technology File 1-4

HFSS 3D Layout Options 1-7

Design Settings 1-10

HFSS 3D Layout Design Settings 1-11

DC Extrapolation Tab 1-12

Export S-Parameters Tab 1-15

HFSS Adaptive Mesh Tab 1-18

HFSS Meshing Method Tab 1-20

Lossy Dielectrics Tab 1-23

Nexxim Options Tab 1-24

Design Mode Tab 1-26

Editing Distributed Machine Configurations in HFSS 3D Layout 1-27

Distributed Machine Configurations for HFSS 3D Layout 1-27

Distributed Machine Configurations for Ansys SIwave™ in HFSS 3D Layout 1-28

Example Projects 1-31

HFSS 3D Layout Example Projects 1-32

Finding Information in the Online Help 1-33

Using the Search Function in the Help 1-33

Help Conventions 1-38

Getting Help from Ansys Technical Support 1-40

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

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

Ansys 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
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
Ansys Electronics Desktop Windows 2-16
Showing and Hiding Windows 2-16
Auto Hiding Windows 2-18
Moving and Resizing Windows 2-18
Project Manager Window 2-21
Project Manager Window for Circuit Designs 2-21
Project Manager Tree Organization 2-22
Closing the Project Manager Window 2-23
Working with the Project Tree 2-23
Setting the Project Tree to Expand Automatically 2-24
Viewing Material Definitions 2-25

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 2-25
Message Manager Window 2-26
Setting the Message Manager to Open Automatically 2-27
Showing New Messages 2-27
Automatically Expanding the Message Manager Tree 2-27
Action Messages 2-28
Clearing Messages 2-28
Hiding the Message Manager Window until Messages Appear 2-28
Progress Window 2-29
Stopping or Aborting Simulation Progress 2-29
Properties Window 2-30
Opening the Properties Window 2-31
Setting the Properties Dialog Box to Open Automatically 2-32
Showing and Hiding the Properties Window 2-32
Modifying Object Attributes Using the Properties Window 2-32
Modifying Object Command Properties Using the Properties Window 2-33
Properties Window for Schematic and Layout Editors 2-33
Param Values Tab 2-33
Launching Help from the Properties Window 2-35
General and Component Tabs 2-35
Symbol Tab 2-36
Properties Dialog Box 2-37
Opening the Properties Dialog Box 2-38
Parameter Values Tab 2-38
Component and Symbol Tabs 2-40
Property Displays Tab 2-41
Working with the Components Window 2-42
Working with the Nets Window 2-43

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

Nets Tab ... 2-44
Classification Pane ... 2-44
Nets Pane ... 2-45
Differential Pairs ... 2-47
Edit Differential Pairs Window ... 2-50
Extended Nets Tab ... 2-51
Extended Nets Pane ... 2-52
Auto Identify Settings window ... 2-55
Nets Pane ... 2-56
Rename ... 2-57
ACT Extensions Window ... 2-60
Working with the Layers Window ... 2-65
Working with the Measure Window ... 2-67
Accessing Measure Mode and Viewing Measured Data ... 2-67
Using Measure mode ... 2-72
Canceling Measurements ... 2-79
Erasing Measurements ... 2-81
Backing Up Measurements ... 2-81
Ansys Electronics Desktop Menus ... 2-82
Menu Bar ... 2-83
File Menu ... 2-85
Edit Menu ... 2-86
Edit Menu for 3D Modeler ... 2-86
Edit Menu for Netlist Editor ... 2-89
Edit Menu for Report Window ... 2-90
Edit Menu for Schematic Editor ... 2-91
Edit Menu for Layout Editor ... 2-92
View Menu ... 2-93

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 ... 2-93
Schematic Editor View Menu ... 2-95
Layout Editor View Menu ... 2-96
3D Modeler View Menu ... 2-99
Project Menu ... 2-105
Editor and Design Specific Menus ... 2-105
Working with Editor Windows ... 2-105
Layout Editor Draw Menu ... 2-107
Tools Menu ... 2-108
Understanding Registry Tools ... 2-111
Adding External Tools to the Tools Menu ... 2-115
Window Menu ... 2-117
Help Menu ... 2-120
Context-Sensitive Help ... 2-121
Obtaining Information about the Software and Release ... 2-121
Shortcut Menus ... 2-122
Shortcut Menu in the 3D Modeler Window ... 2-123
Shortcut Menus in the Project Manager Window ... 2-124
Customizing Ansys Electronics Desktop Menus ... 2-125
Ansys Electronics Desktop Design Area ... 2-134
Report Window ... 2-135
Layout Window ... 2-135
Ansys Electronics Desktop Status Bar ... 2-137
Working with the Status Bar and the Layout Editor ... 2-138
Keyboard Shortcuts ... 2-139
Desktop Shortcuts ... 2-139
Layout Shortcuts ... 2-140
Report Shortcuts ... 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.

Choosing the View Navigation Options ... 2-141
Custom Keyboard Shortcuts ... 2-143
Adding External Tools ... 2-145
Using the Password Manager to Control Access to Resources ... 2-147
Running Ansys Electronics Desktop from a Command Line ... 2-148
Command-Line Syntax ... 2-148
Run Commands ... 2-148
Job Management from the Command Line ... 2-154
Run Command Examples ... 2-155
Specifying Project Files ... 2-155
Options ... 2-156
-Batchoptions ... 2-157
-Batchoptions Examples ... 2-158
Export Options Files ... 2-158
Examples and Further Explanations of -batchoptions Use ... 2-158
For -batchoptions Use: Project Directory and Lib Paths ... 2-161
For -batchoptions Use: TempDirectory ... 2-161
For -batchoptions Use: Various Desktop Settings ... 2-162
Batchoptions: Override Registry Entry ... 2-164
Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta ... 2-165
Command-line Syntax ... 2-165
Run Commands ... 2-166
Run Command Examples ... 2-170
Specifying Project Files ... 2-170
Options ... 2-171
-Batchoptions ... 2-172
-Batchoptions Examples ... 2-172
Export Options Files ... 2-173

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

Running Ansys Electronics Desktop with a JSON File 2-173
JSON File Format 2-173
Running Ansys Electronics Desktop from a Windows Remote Terminal 2-176
Using Windows HPC Commands 2-176
Customizing Ansys Electronics Desktop with Ansys ACT 2-177
Installing PyAEDT (Beta) 2-177
Debug Logging 2-179
3 - Working with Ansys Electronics Desktop Projects 3-1
Ansys Electronics Desktop Files 3-1
Setting Project Options 3-2
Setting General Options 3-3
General Options: Desktop Configuration 3-5
General Options: Project Options 3-7
General Options: Miscellaneous Options 3-8
General Options: User Interface Options 3-9
Choosing a Color Scheme 3-13
General Options: Directories Options 3-15
General Options: Desktop Performance 3-16
General Options: Default Units 3-19
General Options: Remote Analysis Options 3-20
General Options: Component Libraries Options 3-21
Setting Reporter Options 3-23
Reporter Options: Report Setup Options 3-23
Reporter Options: Fields Reporter Options 3-24
Reporter Options: Report2D Options 3-24
Report2D Options: Curve 3-25
Report2D Options: Axis 3-25
Report2D Options: Grid 3-26

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: Header ... 3-26
Report2D Options: Note ... 3-26
Report2D Options: Legend ... 3-27
Report2D Options: Marker ... 3-28
Report2D Options: Marker Table ... 3-28
Report2D Options: X-Y Markers ... 3-28
Report2D Options: Stacked ... 3-30
Report2D Options: Digital ... 3-30
Report2D Options: General ... 3-30
Curve Tooltip ... 3-31
Clipboard Option ... 3-31
Format ... 3-31
Report2D Options: Table ... 3-31
Header Row ... 3-32
Format ... 3-32
Copy to Clipboard ... 3-32
Setting HPC and Analysis Options ... 3-32
Configurations Tab ... 3-33
Selecting an Available Configuration ... 3-34
Options Tab ... 3-35
Specifying the Remote Spawn Command (Linux) ... 3-39
MPI Vendor and MPI Version ... 3-41
Setting Options via Configuration Files ... 3-43
Behavior Examples ... 3-44
Rules for Modifying Option Settings ... 3-44
Configuration File Locations ... 3-44
Products with Multiple Desktop Versions ... 3-45
Table of Directories and Files ... 3-45

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 ... 3-47
UpdateRegistry -Set Command ... 3-47
UpdateRegistry -Get Command ... 3-48
UpdateRegistry -GetKeys Command ... 3-49
UpdateRegistry -Delete Command ... 3-50
UpdateRegistry -FromFile Command ... 3-50
UpdateRegistry File Format ... 3-51
Example Uses for Export Options Features ... 3-52
Options that Apply to All Users ... 3-52
Example: Searching for a Registry Key Pathname ... 3-53
Example: Setting an Installation Default Value ... 3-53
Example: Setting a Host-Dependent Default Value ... 3-53
Example: Reverting from a User-Defined Option Value to the Administrator Default ... 3-54
User Options and the UpdateRegistry Tool ... 3-55
Example: Removing a Host-Dependent User Option Setting ... 3-56
Example: Getting a Value from a Specific Configuration File ... 3-56
Example: Getting a Value Using Precedence Rules ... 3-57
Example: Adding a Host-Independent User Option Setting ... 3-57
Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml ... 3-57
Example: Change a Newly Set Registry Key ... 3-57
Setting the Temporary Directory ... 3-58
Temporary Directory Configuration File Format ... 3-60
UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files ... 3-61
Batchoptions Command Line Examples ... 3-62
Batchoptions File Format ... 3-62

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

Example -BatchOptions with -Remote (Windows) 3-63
Example -Batchsolve with -Machinelist (Windows) 3-64
Example -Batchsolve with -Machinelist (Linux) 3-64
Example -Batchsolve for Local (Windows) 3-65
Batch Options and Analysis Configurations in the Registry 3-66
Managing Projects and Designs 3-70
Opening Projects 3-71
Opening Recent Projects 3-73
Opening Example Projects 3-74
Opening Legacy HFSS-IE Projects 3-75
Opening Legacy Projects 3-75
Legacy Project Translation 3-76
Creating Projects 3-77
Saving Projects 3-79
Path Name Length Issues for Windows 3-80
Changing Auto-Save Settings 3-81
Save Before Solving Option 3-82
Recovering Project Data from an Auto-Save File 3-82
Choosing a Preview Image 3-83
Importing and Exporting Projects and Data 3-85
Copying and Pasting a Project or Design 3-86
Renaming Projects 3-86
Archiving Projects 3-86
Restoring an Archived Project 3-90
Deleting a Project or Design 3-93
Setting Read Only Designs 3-93
Updating Design Components 3-94
Undoing and Redoing Commands 3-95

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 ... 3-96
Modeler Validation Settings ... 3-97
Inserting a Documentation File ... 3-98
Working with Design Notes ... 3-98
Printing ... 3-100
Closing Projects ... 3-101
Example Projects ... 3-101
HFSS 3D Layout Example Projects ... 3-102
Exporting Options Files ... 3-104
Exporting Options Files Using the Desktop UI ... 3-104
Exporting Options Files Using a Script ... 3-104
Working with Variables ... 3-104
Working with Project Variables ... 3-106
Adding a Project Variable ... 3-106
Importing and Exporting Project Variables ... 3-109
Deleting a Project Variable ... 3-110
Editing a Project Variable ... 3-110
Adding a Project Variable Array ... 3-111
List of Intrinsic Variables ... 3-115
List of Constants ... 3-116
Working with Design Variables ... 3-117
Defining Local Variables at the Design Level ... 3-118
Defining Parameter Defaults at the Design Level ... 3-123
Adding a Design Variable Array ... 3-125
Converting Variables and Parameter Defaults ... 3-131
Adding Variables and Parameter Defaults for Components ... 3-132
Fixed vs. Non-Fixed Variables ... 3-135
Non-Fixed Variable Sweeps ... 3-135

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

Orphaned Sweeps ... 3-136
Choosing a Variable to Optimize ... 3-137
Choosing a Variable to Tune ... 3-138
Including a Variable in a Sensitivity Analysis ... 3-139
Including a Variable in a Statistical Analysis ... 3-140
Copying and Pasting a Variables List ... 3-141
Working with Datasets ... 3-142
Adding Datasets ... 3-142
Importing Datasets ... 3-146
Editing Datasets ... 3-148
Cloning Datasets ... 3-150
Exporting Datasets ... 3-152
Removing Datasets ... 3-152
Changing Dataset Plot Properties ... 3-154
Using SheetScan ... 3-156
SheetScan Menus and Settings ... 3-156
SheetScan View Options ... 3-157
SheetScan Right-click Menu ... 3-158
SheetScan Settings ... 3-158
Loading a Datasheet Picture into SheetScan ... 3-158
Loading a Datasheet Picture Directly ... 3-159
Loading a Datasheet Picture Using the HTML Viewer ... 3-159
Defining a SheetScan Coordinate System ... 3-159
Defining a Characteristic Curve in SheetScan ... 3-161
Performing Operations on SheetScan Curves ... 3-162
Checking for Monotonicity in X ... 3-163
Importing Characteristic Data into SheetScan ... 3-163
Exporting SheetScan Data ... 3-165

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

Defining an Expression ... 3-166
Defining Mathematical Functions ... 3-167
Valid Operators for Expressions ... 3-167
Using Intrinsic Functions in Expressions ... 3-168
clp Formula ... 3-170
Using Piecewise Linear Functions in Expressions ... 3-170
Using Dataset Expressions ... 3-171
Cosimulation of Circuit and Field Solver Designs ... 3-173
4 - Minerva Remote Storage Environment ... 4-1
5 - Layout and Circuit Import Export Operations ... 5-1
Importing Layout Design Data ... 5-1
Importing ANF Design Data ... 5-1
Importing ANFV2 Design Data ... 5-2
Importing ANFV4 Design Data ... 5-3
Importing Files into the Layout Editor ... 5-5
Importing EDB Format Files ... 5-6
Importing Gerber Format Files into the Layout Editor ... 5-8
Importing IDF Files ... 5-9
Overview of Importing IDF Files ... 5-10
Import an IDF File ... 5-10
Importing ODB++ into the Layout Editor ... 5-11
Importing IPC2581 rev.a, rev.b, and rev.c Files into the Layout Editor ... 5-13
Importing DXF and DWG Format Files into the Layout Editor ... 5-14
Importing Cadence APD/ALLEGRO/SIP Files into the Layout Editor ... 5-16
Importing XFL Files into the Layout Editor ... 5-18
Importing GDSII Format Files in IC Mode ... 5-18
Importing a GDSII file ... 5-18
The Preview Stackup Window ... 5-24

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

The Cell Summary Window ... 5-25
Reset ... 5-26
Adding Ports From a .CSV File ... 5-26
Net Tracing in IC Mode ... 5-30
Viewing Physically Connected Elements in IC Mode ... 5-31
Importing Tabular Arrays ... 5-37
Direct Import Cadence APD Links ... 5-40
Layout Links ... 5-40
Cadence APD Links ... 5-41
Configure Ports for Dynamic Interaction with the Cadence Data Source ... 5-41
Configure Import Options ... 5-41
Configure and View Active Bondwire Definitions ... 5-41
SPB License For Direct Import ... 5-41
Prerequisites ... 5-41
Running AnsTranslator on the Command Line ... 5-42
Net Importation in AnsTranslator ... 5-44
Layout and Control File ... 5-45
Control File Usage with Renaming ... 5-49
Using Excel to Generate a Control File ... 5-52
Exporter Issues ... 5-58
Layer Mapping and XML Control Files ... 5-65
GDS Translation Using XML Control Files ... 5-68
Stackup ... 5-69
LayerAttr ... 5-70
ConvertPolygonToCircle ... 5-70
ViaLayer ... 5-70
TSVProperties ... 5-70
Geometry ... 5-70

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

PolygonWithHoles ... 5-71
PreCutout ... 5-71
CriticalNet ... 5-72
CriticalNetGroup ... 5-72
RemoveHoles ... 5-73
Boundaries ... 5-73
CircuitPortPt ... 5-73
Extents ... 5-73
BoundaryConditions ... 5-74
Components ... 5-74
COMPONENTPORT ... 5-74
SPICEMODEL ... 5-75
CutoutSubdesign ... 5-76
SimulationSetups ... 5-76
HFSSSetup ... 5-76
HFSSSimulationSettings ... 5-76
HFSSSweepDataList ... 5-77
RemoveIslandsTolerance ... 5-77
ImportOptions ... 5-77
ImportOptions ... 5-78
GDS_CELL_RULES ... 5-78
GDS_NET_DEFINITIONS ... 5-79
Auto Net Tracing in GDS/OASIS Designs ... 5-80
Configuration ... 5-80
GDS_COMPONENTS ... 5-81
GDS_TEXT_PINS ... 5-82
GDS_AUTO_COMPONENT ... 5-84
AUTOCOMPONENTPORT ... 5-84

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

ODB++ Translator ... 5-85
Importing ODB++ in Ansys Electronics Desktop ... 5-85
Control File Part Mapping ... 5-87
Exporting Layout Design Data ... 5-90
Exporting Data Table Files ... 5-90
Exporting Layout Data ... 5-91
DXF Export ... 5-93
GDSII Export ... 5-93
Gerber Export ... 5-95
NC Drill/Router Export ... 5-96
Exporting Graphics Files on the Layout 3D Editor ... 5-97
Exporting ANF Layout or Schematic Data ... 5-98
Importing Circuit Models ... 5-98
Selecting a Model to Import ... 5-101
SPICE Library Support ... 5-103
SPICE Library Import Procedure ... 5-104
PSPICE Library Import Procedure ... 5-106
Modifications to SPICE Pin Names ... 5-108
PSPICE Model Support ... 5-109
Semiconductor Model Support ... 5-109
PSPICE Function Support ... 5-110
PSPICE Vendor Library Support ... 5-112
Coilcraft Power Inductors ... 5-112
IBIS Library Support ... 5-114
Import IBIS Library File ... 5-115
Import IBIS Single Component or Pin or Buffer Element ... 5-117
Import Single Component with [Package Model], All Pins ... 5-117
Pin Import Tab ... 5-124

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

Buffer Import Tab ... 5-127
Place IBIS Buffer Component in Schematic ... 5-128
Edit IBIS Component Properties ... 5-133
IBIS Property Locations ... 5-133
Basic Buffer Properties ... 5-135
Component and Pin Import Properties ... 5-137
Package Support Properties ... 5-141
AMI Specific Properties ... 5-142
IBIS File Import Technical Notes ... 5-142
Support for IBIS Specification ... 5-143
Power Nodes ... 5-143
Package RLC Parasitics ... 5-144
Support for Power Aware IBIS Output Buffer Models ... 5-145
Digital Voltage Levels ... 5-149
Input Node Thresholds ... 5-149
Output Node Thresholds ... 5-150
Enable Node Thresholds ... 5-151
Transition Settling Times ... 5-151
Notes on IBIS Waveform Properties ... 5-152
TYP Parameter Effects ... 5-155
Importing IBIS EBD Files ... 5-156
IBIS I/O Wizard ... 5-158
IBIS I/O Wizard window ... 5-158
IBIS I/O Wizard Signal Setup ... 5-159
IBIS I/O Wizard Power/Ground Pin Setup ... 5-160
IBIS I/O Wizard Source/Analysis Setup ... 5-162
AMI Analysis ... 5-162
Set Up and Run AMI Analysis ... 5-163

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

AMI Analysis Outputs ... 5-163
W-Element Model Support ... 5-163
X-Parameter Model Support ... 5-165
Importing an X-Parameter Component ... 5-165
X-Parameter Technical Notes ... 5-168
Independent Variables for X-Parameters ... 5-173
X-Parameter References ... 5-174
State Space Model Support ... 5-174
Importing State-Space Models ... 5-175
Editing a State Space Model ... 5-176
State Space N-Port Data Source Tab ... 5-176
State Space N-Port Data Noise Data Tab ... 5-177
State Space N-Port Data Options Tab ... 5-178
Editing State Space N-Port Symbol Pins ... 5-181
Creating a Multi-File State Space Model ... 5-181
Editing a Multi-File State Space Model ... 5-182
Importing Network Design Data (SYZ-Parameters) ... 5-183
Creating an N-Port Model ... 5-184
Editing an N-Port Model ... 5-185
Creating a Multi-File N-Port Model ... 5-185
Editing a Multi-File N-Port Model ... 5-187
Creating an N-Port Model with No File ... 5-188
Creating a Parametric N-Port Model ... 5-190
Editing a Parametric N-Port Model ... 5-192
Parametric N-Port Data Source Tab ... 5-192
Parametric Link Description Tab ... 5-193
Parametric Options Tab ... 5-196
Parametric Simulation Tab ... 5-196

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

Editing Parametric N-Port Symbol Bus Pins ... 5-199
Touchstone Calibration Wizard ... 5-199
Two-Line Method ... 5-199
Thru-Reflect-Line Method ... 5-200
Short-Open-Load-Thru Method ... 5-201
Paths for File Selection ... 5-201
Editing All N-Port Symbol Bus Pins ... 5-202
Verilog Library Support ... 5-204
Create a Verilog Component ... 5-208
Supported Verilog Features ... 5-213
CPM Model Support ... 5-217
Automated CPM Chip Connection And Placement ... 5-218
Exporting Network Design Data ... 5-218
Exporting LNA Network Data ... 5-218
Exporting Network Data to a State Space File ... 5-220
Importing and Exporting Netlist Data ... 5-221
Accessing Netlist Files ... 5-221
Include Files in Circuit Netlists ... 5-223
Accessing Include Files ... 5-223
Library Files in Circuit Netlists ... 5-226
Accessing Library Files ... 5-226
Exporting Netlist Data ... 5-229
Importing Solution Data for Postprocessing ... 5-230
Importing Network Solution Data ... 5-230
N-Port Data Tab for Imported Solutions ... 5-231
Noise Data Tab for Imported Solutions ... 5-234
State Space N-Port Data Options Tab ... 5-236
Importing Tabular Solution Data ... 5-238

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

Post-processing Imported Solution Data 5-242
6 - Geometry in HFSS 3D Layout 6-1
Stride Lightweight Geometry in HFSS 3D Layout 6-1
Importing 3D CAD Components into HFSS 3D Layout 6-2
Exporting HFSS 3D Layout to Stride Format 6-6
Adding HFSS 3D Components to HFSS 3D Layout 6-8
3D Component Library 6-9
Using the Component Libraries Window 6-11
Component File Organization on the Disk 6-14
Viewing Properties of 3D Components in 3D Layout 6-14
3D Components in the Project Manager Window of HFSS 3D Layout 6-15
Editing 3D Component Properties 6-18
Editing a 3D Component Definition 6-21
Verifying Updates to Components 6-25
Updating Definitions 6-25
Replacing 3D Components 6-26
Stackup Definitions for HFSS 3D Models 6-27
Substrate Stackup 6-27
HFSS Etch Factor 6-29
Layout Stackup 6-29
Configuring a Stackup 6-30
Laminate Stackup 6-31
Multizone Stackups and Rigid-Flex Bends 6-31
Creating or Importing a Multizone Stackup 6-32
Importing a Multizone Stackup 6-32
Creating a Multizone Stackup 6-34
Creating or Copying Zones 6-36
Adding a New Zone 6-38

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 New Zone With Specific Layers 6-39
Copying a Zone 6-40
Editing Zones 6-41
Defining Zones 6-44
Understanding Zone Boundaries 6-50
Using HFSS Model Extents to Modify Zones 6-50
Using the Implied Regions to Assign Zones 6-55
Changing the Zone Order 6-56
Reorienting the Model and Stretching Z 6-58
Defining Zones at the Layout Editor 6-61
Defining Rigid-Flex Bends 6-63
Bend Restrictions 6-70
Setting a Defeaturing Tolerance (HFSS > Advanced Setup Only) 6-73
Removing Voids (HFSS > Advanced Setup Only) 6-73
Defining Bends at the Layout Editor 6-74
Animating Rigid-Flex Bends 6-77
Analyzing a Design With Rigid-Flex Bends 6-81
Deactivating Simplified Bend Graphics 6-84
Exporting/Importing a Flexed Design Component 6-86
Exporting a Flexed Design Component 6-87
Importing a Flexed Design Component 6-93
Overlapping Stackup 6-96
Footprint Stackup 6-96
Model a Signal Layer as a Ground Plane 6-97
Subcircuits with Independent Stackups 6-98
Using the Layer Stackup Wizard (HFSS 3D Layout) 6-99
Opening the Layer Stackup Wizard 6-99
Configuring the Layer Stackup 6-100

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

Exporting Data ... 6-101
Viewing Impedance Plots ... 6-101
Other Capabilities of the Layer Stackup Wizard ... 6-102
Calculating Trace Width and Pitch Using the Layer Stackup Wizard (HFSS 3D Layout) ... 6-102
Calculating Impedance Using the Layer Stackup Wizard (HFSS 3D Layout) ... 6-107
Calculating Differential Impedance Using the Layer Stackup Wizard (HFSS 3D Layout) ... 6-108
Exporting the W-element from Layer Stackup Wizard (HFSS 3D Layout) ... 6-110
Computing Insertion/Return Loss and Time Domain Crosstalk (HFSS 3D Layout) ... 6-112
Adding Programmatic EDB Cells to HFSS 3D Layout ... 6-115
Creating a Component ... 6-115
Editing the Python Script ... 6-116
Saving to Libraries ... 6-116
Component Instances ... 6-116
Setting Uniform Temperature in HFSS 3D Layout ... 6-116
7 - Layout Editor ... 7-1
IC Mode ... 7-1
IC Mode Workflow ... 7-1
Switching to IC Mode ... 7-2
Switching to IC Mode from the HFSS 3D Layout Menu ... 7-4
Switching to IC Mode from the Properties Window ... 7-6
Opening a Design in IC Mode ... 7-8
Layout Editor Display and Menus ... 7-9
Layout Editor Display ... 7-9
Layout Editor Options ... 7-10
Setting Global Layout Options ... 7-10
Layout Editor Options: Object Panel ... 7-11

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 Options: Snapping Panel 7-13
Layout Editor Options: Display Panel 7-14
Layout Editor Options: Selection Panel 7-15
Layout Editor Options: Other Panel 7-16
Changing Naming Conventions 7-17
Example Naming Convention - Port Instance Port 7-19
Layout Editor Options: Advanced Panel 7-20
Setting Local Layout Options 7-20
Edit Menu 7-21
IC Mode Commands 7-23
View Menu 7-24
IC Mode Commands 7-25
Draw Menu 7-25
IC Mode Commands 7-27
Drawing a Bondwire in HFSS 3D Layout 7-28
Bondwire Definitions 7-29
Add a Bondwire Definition 7-30
Editing a Bondwire Definition 7-33
Importing Cadence Bondwire Profiles 7-35
Cloning a Bondwire Definition 7-35
Removing a Bondwire Definition 7-35
Export Bondwire Definitions to File 7-35
Drawing a Void in the Layout Editor 7-35
Merge Polygons 7-38
Pin Groups in 3D Layout 7-39
Creating a Pin Group 7-39
Editing a Pin Group 7-43
Renaming a Pin Group 7-44

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 Pin Group 7-44
HFSS 3D Layout Properties 7-44
Meshing Properties 7-44
Modeling Props for 3D Structures 7-47
Precedence Prioritizing 7-47
Thin Film Properties 7-50
Selecting the Sweep Type 7-50
Generating the Surface Current 7-51
Setting up a Full-Wave SPICE Analysis 7-51
Guidelines for Calculating Frequencies for a Full-Wave SPICE Analysis 7-52
Clip Planes 7-53
Convert Traces to Planes 7-54
Convert Planes to Traces 7-54
Drawing a Cross-Section for W-element Analysis (HFSS 3D Layout) 7-55
Layout Menu 7-58
IC Mode Commands 7-60
The Design List 7-61
Layout Tab 7-62
Nets Tab 7-63
Net Classes Tab 7-64
Extended Nets Tab 7-65
Nets 7-66
General Mode Only 7-71
Cutout Subdesign 7-72
Line Styles 7-74
Legacy Line Style Attributes 7-74
Specifying Line Ending Types 7-75
Text Styles 7-75

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

System and Plotter Layers 7-75
System Layers 7-75
Plotter Layers 7-75
Stroke and TrueType Fonts 7-75
Scalable and Nonscalable Fonts 7-76
Font Requirements for System and Plotter Layers 7-76
Layout Editor Context Menu 7-76
IC Mode Commands 7-79
Shortcut Ribbon (IC Mode) 7-79
Shortcut Ribbon Buttons - No Selection 7-80
Shortcut Ribbon Buttons - Selection 7-81
Layout Windows 7-82
Merge Layers Window 7-82
Working with the Components Window 7-83
Place Design Window 7-84
Pin Connectivity Window 7-87
Basic Usage 7-87
Controls 7-87
Configuration, shortcuts, and additional notes 7-88
Solver Options Window 7-89
The Edit Layers Window 7-90
Stackup Type 7-92
Display Options and Units 7-92
Grid Control Pane 7-93
Layer Pane 7-95
Edit Selected Pane 7-95
Stackup Viewer 7-96
Color, Visibility, and Patterns 7-97

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

Setting a Layer Color 7-97
Setting a Layer Pattern 7-97
Setting Layer Visibility 7-97
Layout Technology 7-98
Choosing a Layout Technology 7-98
Creating a Layout Technology 7-102
The Geometry Simplification Settings Window (IC Mode Only) 7-102
Opening the Geometry Simplification Settings Window 7-103
Via Snapping 7-104
Primitive Snapping 7-104
Via Grouping 7-104
Geometry Wrapping 7-105
Remove Small Metal Islands 7-105
Reconstruct Arcs 7-105
Remove Small Lines 7-105
Remove Small Fragments 7-105
Snapping Vias in IC Mode 7-105
Configuring Via Snapping Settings 7-106
Snapping Vias on a Specific Layer 7-108
Snapping Vias in the Layout Editor 7-112
Grouping Vias in IC Mode 7-113
Group By Density 7-114
Group By Proximity 7-114
Group By Range 7-114
Configuring Via Grouping Settings 7-114
Grouping Vias on a Specific Layer 7-117
Manually Grouping Vias in the Layout Editor 7-121
Uniting Primitives in IC Mode 7-122

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

Reconstructing Arcs in IC Mode 7-125
Configuring Reconstruct Arcs Settings 7-125
Reconstructing Arcs in the Layout Editor 7-127
Wrapping Geometry in IC Mode 7-127
Snapping Primitives in IC Mode 7-130
Configuring Primitive Snapping Settings 7-130
Snapping Primitives in the Layout Editor 7-132
Snapping Primitives on a Specific Layer 7-134
Snapping Primitives On Visible Layers 7-135
Removing Objects and Layers in IC Mode 7-136
Trimming Layers from an Encrypted Design 7-138
Removing Small Islands in IC Mode 7-140
Configuring Remove Small Islands Settings 7-141
Removing Small Islands in the Layout Editor 7-144
Removing Small Lines in IC Mode 7-145
Configuring Remove Small Lines Settings 7-146
Removing Small Lines in the Layout Editor 7-148
Removing Small Holes in IC Mode 7-151
Accessing the Remove Small Holes Window 7-152
Removing Small Holes Using Area Threshold 7-153
Removing Small Holes Using Net Exclusion 7-155
Expanding the Exclusion Area 7-158
Cropping and Removing Small Fragments in IC Mode 7-159
Cropping a Design 7-159
Removing Small Fragments From a Design 7-163
Copying and Pasting in IC Mode 7-166
Working with Objects 7-168
Selection Modes in the Layout Editor 7-168

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

Using Handles 7-169
Selecting Handles 7-170
Aligning Objects in the Layout Editor 7-170
Selecting Overlapping Objects in the Layout Editor 7-172
Selecting Edges in the Layout Editor 7-172
Grouping Objects in the Layout Editor 7-172
Negative Objects in the Layout Editor 7-173
Position Locking 7-173
Modifying Objects in the Layout Editor 7-174
Deleting Objects in the Layout Editor 7-174
Flipping Objects in the Layout Editor 7-174
Rotating Objects in the Layout Editor 7-174
Uniting Objects in the Layout Editor 7-175
Subtracting Objects in the Layout Editor 7-176
Creating Objects from Intersections in the Layout Editor 7-176
Duplicating Objects in the Layout Editor 7-177
Expanding Objects in the Layout Editor 7-178
Paste Objects In Place in the Layout Editor 7-179
Geometry Healing in the Layout Editor 7-179
Point Snapping 7-181
Feature Cleanup 7-183
Repair 7-185
Editing Arcs in the Layout Editor 7-186
Adding Vertices to Edges in the Layout Editor 7-186
Working with Layers 7-187
Working with the Layers Window 7-188
Activating Layers 7-190
Layers Workspace 7-190

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

Edit Layers Window 7-190
Layout Editor Ribbon 7-191
Adding and Removing Layers 7-192
Adding a Layer 7-192
Removing a Layer 7-193
Working With TSV Layers 7-194
Active Design With Oxide Layers 7-196
Active Design Without Oxide Layers 7-196
Assigning Materials to Layers 7-197
Default Layers 7-197
Dielectric Simplification (Merging) 7-197
Drawing on a Layer 7-200
Setting Layer Associations 7-200
Setting Layer Order and Elevations 7-201
Elevation in Laminate and Multizone Stackups 7-201
Elevation in Overlapping Stackups 7-201
Method A 7-201
Method B 7-202
Setting the Layer Lower Elevation 7-202
Setting the Layer Upper Elevation 7-202
Setting Layer Thickness 7-202
Setting the Etch Factor 7-203
Accessing the HFSS Etch Factor Window 7-204
Entering a New Etch Factor 7-206
The Calculator Area 7-208
The Etching By Net Class Area 7-208
Setting the Layer Surface Roughness 7-209
Groiss Surface Roughness Model 7-210

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

Huray Surface Roughness Model 7-211
Working with Dielectric and Negative Signal Layers 7-213
Via Layers 7-213
Drawing a Model in the Layout Editor 7-214
The Drawing Region 7-214
General Drawing Guidelines for the Layout Editor 7-215
Drawing Apertures on Negative Signal Planes 7-215
Zooming and Panning the Layout View 7-215
Cross-Probing Selected Elements 7-217
Voids in Layout 7-217
Directional Drawing in the Layout Editor 7-218
Drawing Geometric Objects in the Layout Editor 7-219
Drawing a Circle in the Layout Editor 7-219
Drawing a Rectangle in the Layout Editor 7-219
Drawing an Image in the Layout Editor 7-220
Drawing an Arc in the Layout Editor 7-221
Additional Notes for Drawing an Arc 7-221
Drawing a 3D Line Segment 7-222
Drawing a Line in the Layout Editor 7-230
Specifying Line Width in the Layout Editor 7-231
Reverse Line Command in the Layout Editor 7-232
Drawing a Polygon in the Layout Editor 7-232
Curving Polygon Edges in the Layout Editor 7-233
Drawing a Void in the Layout Editor 7-233
Drawing a Via in the Layout Editor 7-235
Drawing a Cross-Layer Plate in the Layout Editor 7-236
Drawing a Hole in the Layout Editor 7-236
Drawing a 2.5D Via in the Layout Editor 7-237

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

Selecting Load Types for 2.5D Vias 7-241
Creating Vias, Pins, and Padstack Definitions from Primitives 7-242
To Create Vias from Primitives: 7-242
To Create Pins from Vias: 7-245
To See the Geometry of a New Via: 7-245
Compressing Primitives in the Layout Editor (IC Mode Only) 7-245
Via Clustering (Via Groups) 7-248
Inserting N-Port Data 7-249
Guidelines for Inserting N-Ports 7-251
Importing or Linking N-Port Data 7-252
Manually Entering N-Port Data 7-253
Changing the Reference Impedance for N-Port Data 7-254
Rotating the Phase of N-Port Data 7-254
Drawing a Connection in the Layout Editor 7-254
Components with a Symbol Footprint 7-256
Assigning Reference Ports in the Layout Editor 7-256
Circuit Ports in Layout 7-258
Dielectric Modeling 7-259
Importing a Stackup 7-260
Components 7-262
Creating a Component 7-263
Editing a Component 7-266
SPICE Model 7-270
Manual Model 7-273
Die Properties 7-274
Solder Ball Properties 7-275
Port Properties 7-275
Creating or Editing Package Definitions for Components 7-275

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

Intersection Checks for 3D Components ... 7-279
Creating a Mesh Region to Validate Encrypted Designs ... 7-285
Deactivating the Intersection Check ... 7-285
Next Steps ... 7-288
Creating a Circuit Element ... 7-288
Modifying a Circuit Element's Properties ... 7-290
Connecting a Circuit Element ... 7-291
Auto Scale of Circuit Elements ... 7-293
Turn Off Auto Scale ... 7-295
Nport Circuit Elements ... 7-296
Creating a Circuit Element Between Any Two Layers ... 7-298
Physical Definition for EM Models ... 7-302
User-Defined Components ... 7-302
Library Components ... 7-302
Working With The Capacitor/Inductor Library Browser ... 7-303
Add a New Member to a Component ... 7-305
Remove a Member from a Component ... 7-305
Dissolve a Component ... 7-307
Importing and Exporting Layout Components ... 7-307
Importing a Layout Component (General Mode Only) ... 7-317
Visible Restrictions of an Encrypted Layout Component ... 7-319
Layout Editor ... 7-319
Layout Editor - Mesh View ... 7-320
Components Window ... 7-321
Properties Window ... 7-322
Edit Layers Window and Layers Pane ... 7-324
Hiding Layers and Nets ... 7-325
To Hide a Layer ... 7-325

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

To Hide a Net ... 7-326
Exporting a Layout Component ... 7-327
Editing a Layout Component Definition ... 7-337
Replacing Layout Components ... 7-349
Encrypted Designs (IC Mode) ... 7-350
Adding Layers to an Encrypted Design ... 7-350
Enabling Solver Options in Encrypted Layers ... 7-352
Creating Instance Arrays in IC Mode ... 7-354
Selecting Instance Array ... 7-355
Navigating a Hierarchical Design ... 7-357
Dissolving an Instance Array ... 7-357
Creating Circuit Ports in IC Mode ... 7-359
Creating a Point-to-Point Circuit Port ... 7-359
Editing Circuit Ports ... 7-363
3D Structures ... 7-366
Creating 3D Structures ... 7-367
3D Ungroup ... 7-367
3D Push Down and Pop Up ... 7-367
Hierarchical Design ... 7-369
Coordinate Systems ... 7-369
Creating a Coordinate System ... 7-370
New Coordinate System with No Object Selected ... 7-370
New Coordinate System with Edge, Port, or Port Instance Selected ... 7-370
New Coordinate System with a Non-Applicable Object Selected ... 7-371
Selecting a Coordinate System ... 7-371
Push Down and Pop Up Commands ... 7-371
DC IR and Icepak Simulation on Hierarchical Designs ... 7-373
SIwave Export to Electronics Desktop ... 7-373

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

DC IR Simulation on Hierarchical Designs 7-374
Before – Sources are spread across two levels of the hierarchical design. 7-379
After – Sources are shown on the top level of the hierarchical design. 7-380
Two-Way Coupled Icepak Simulation 7-382
Coordinate Systems and Hierarchy Navigation 7-382
Validation and Visualization 7-383
Activating the ECAD-ECAD Hierarchical Design Beta Feature 7-383
Mesh Fusion/Assembly for Hierarchical Design 7-386
Mesh Method Override 7-388
Access the Mesh Settings Window 7-388
Mesh Fusion/Assembly for Hierarchical Design 7-390
Mesh Method Override 7-392
Access the Mesh Settings Window 7-393
Geometry Check 7-395
Printing a Layout 7-396
8 - Assigning Materials 8-1
Solving Inside or on the Surface 8-1
Assigning DC Thickness 8-1
Searching for Materials 8-4
Searching by Material Property 8-6
Adding New Materials 8-7
Assigning Material Property Types 8-10
Defining Anisotropic Tensors 8-11
Assigning Anisotropic Tensors 8-11
Defining Variable Material Properties 8-12
Defining Frequency-Dependent Material Properties 8-12
Assigning Frequency-Dependent Properties 8-13
Frequency Dependence Visualization 8-15

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

Saved Input Data Invalidation For Frequency-Dependent Setup 8-16
Assigning Frequency Dependent Material: Piecewise Linear Input 8-16
Assigning Frequency Dependent Material: Debye Model Input 8-18
Assigning Frequency Dependent Material: Multipole Debye Model Input 8-19
Assigning Frequency Dependent Material: Djordjevic-Sarkar Model Input 8-22
Assigning Frequency Dependent Material: Enter Frequency Dependent Data Points 8-23
Specifying Thermal Modifiers 8-25
Defining Material Properties as Expressions 8-32
Defining Functional Material Properties 8-33
Assigning Materials from the Object Properties Window 8-33
Viewing and Modifying Material Attributes 8-33
Validating Materials 8-36
Copying Materials 8-36
Removing Materials 8-36
Exporting Materials to a Library 8-37
Sorting Materials 8-37
Filtering Materials 8-37
Working with Material Libraries 8-39
Editing Libraries 8-39
9 - Port Setup in HFSS 3D Layout 9-1
HFSS 3D Layout Port Types 9-1
Ports for Transmission Lines 9-2
Creating Edge Ports 9-2
Creating PEC Edge Ports 9-9
Creating RLC Edge Ports 9-9
Changing the Type of Port or Boundary 9-10
Single-Line Transmission Line Ports 9-10

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

Multi-Line Transmission Line Ports 9-11
Creating Coupled Ports From Existing Edge Ports 9-12
Removing an Edge Port or Boundary 9-12
Ports for Imported Package and PCB Structures 9-13
Creating Pin-Based Ports 9-13
Configuring Pin-Based Ports 9-13
Creating a Pad Port 9-14
Configuring Ports 9-15
Gap(coax) Ports 9-28
Terminal Assignments in Gap(coax) Ports 9-28
Wave(coax) Ports 9-29
Horizontal Wave Ports 9-31
Horizontal Lumped Ports 9-34
CPW and Slotline Ports 9-40
Circuit Ports and Elements in HFSS 3D Layout 9-41
Circuit Element Advantages 9-41
Creating a Circuit Port 9-46
Interface Ports and Sources in 3D Layout 9-49
Changing the Reference Surface 9-52
Selecting a Dual Reference 9-52
Reference to Nearest Negative Signal 9-53
Configuring Multiple Ports 9-53
Validating Connectivity and Return Paths 9-54
Exposing Horizontal Lumped Ports in Hierarchy 9-55
Port Settings for Post Processing 9-55
Edit Port Excitations 9-56
Renormalizing S-Matrices 9-59
Deembedding S-Matrices 9-61

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

Varying the Port Excitation ... 9-63
Differential Pairs ... 9-65
Pushing Excitations ... 9-65
Push Excitations to HFSS 3D Layout Ports ... 9-65
Push Excitations to Dynamic Links ... 9-70
Push Excitations to Sub-designs ... 9-70
Push Excitations with Thevenin Equivalent ... 9-71
10 - Solution Setup in HFSS 3D Layout ... 10-1
Adding a Solution Setup for HFSS 3D Layout ... 10-1
Adding an HFSS Solution Setup (General Mode and IC Mode) ... 10-1
Adding an Auto HFSS Solution Setup ... 10-3
Adding an Advanced HFSS Solution Setup ... 10-5
Setting the Solution Frequency ... 10-10
Setting the Maximum Number of Passes ... 10-13
HFSS Setup: General Tab ... 10-14
The Matrix Convergence Window ... 10-19
Activating the Parallel Refinement Beta Feature ... 10-25
Enable Auto Select Direct/Iterative ... 10-32
Enable the Auto Select Direct/Iterative ... 10-32
Enable Direct Solver ... 10-34
Enable the Direct Solver ... 10-34
Enable Iterative Solver ... 10-34
Enable the Iterative Solver ... 10-34
HFSS 3D Model Extents ... 10-45
Define or Edit HFSS 3D Model Extents ... 10-45
View or Hide the Extents ... 10-55
Adding an HFSS-PI Solution Setup ... 10-55
Setting Up an HFSS-PI Simulation ... 10-55

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

Analyzing the Design 10-62
Post-Processing with HFSS-PI Simulation Setups 10-62
Activating the Enhanced PI Sweep Beta Feature 10-67
Adding a RaptorX Solution Setup (IC Mode Only) 10-70
Setting Up a RaptorX Simulation 10-70
Changing Net Settings 10-79
Applying Technology 10-81
Analyzing the Design and Post-Processing 10-83
Viewing Green's Function Data 10-89
Generating Green's Function Data 10-92
Adding a Q3D Solution Setup 10-93
Viewing RLGC Matrix Data 10-129
Changing Net Settings 10-131
Editing Sources 10-134
Exporting RLGC Matrix Data 10-137
Exporting RLGC Data in Several Formats 10-139
Exporting Lumped RLGC Data in an Equivalent Circuit File Format 10-142
Activating the Q3D Solution Setup Beta Feature 10-148
Ansys SIwave™ Solutions in HFSS 3D Layout 10-151
SIwave Workflow Wizard in HFSS 3D Layout (General Mode Only) 10-152
Assigning SPICE/S-parameter Vendor Models in HFSS 3D Layout 10-155
Attaching a SPICE RLC Model to a Component in HFSS 3D Layout 10-158
Computing SYZ Solutions in HFSS 3D Layout (General Mode Only) 10-162
To Compute the S-,Y-, and Z-Parameters 10-163
Ansys SIwave™ Region-Based Simulations in HFSS 3D Layout 10-177
Methods of Defining HFSS Regions 10-178
HFSS Regions for Ansys SIwave Sweeps 10-179
Rational Interpolation 10-182

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

Hierarchical Restrictions 10-183
Generating a Regions Schematic in HFSS 3D Layout 10-186
Configuring a Parallel HFSS Regions Simulation in HFSS 3D Layout 10-188
Example 10-189
Generating Automatic HFSS Regions for Ansys SIwave™ (Beta) 10-190
Viewing and Exporting SYZ Solution Results 10-192
Configuring S-Parameters in an Ansys SIwave™ Solution Setup (Example) 10-193
Computing Near Field Solutions in HFSS 3D Layout (General Mode Only) 10-198
Setting Up a Near Field Simulation 10-198
Setting Up a Frequency Sweep 10-205
Assigning a Transient Excitation 10-210
Analyzing the Design and Post-Processing 10-212
Using Two X Markers 10-222
Computing Far Field Solutions in HFSS 3D Layout (General Mode Only) 10-224
Setting Up a Far Field Simulation 10-225
Setting Up a Frequency Sweep 10-230
Assigning a Transient Excitation 10-234
Analyzing the Design and Post-Processing 10-237
Using Two X Markers 10-240
Computing DC IR Simulations in HFSS 3D Layout 10-243
Computing DC IR Simulations in IC Mode 10-245
Beginning Simulation From the SIwave Workflow Wizard (General Mode Only) 10-245
Setting Up the Compute DC Current and Voltage Distribution Window 10-251
Assigning a Transient Excitation 10-261
Adding Sources, Probes, and Terminals 10-265
Adding Voltage Sources, Current Sources, and Voltage Probes 10-265
Creating Sources on Components 10-271

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

Defining Equipotential Regions (General Mode Only) 10-272
Editing the DC Coefficient Text File 10-274
Viewing and Creating VRMs (General Mode Only) 10-274
Viewing and Exporting DC IR Solution Results 10-280
Display Element Data 10-282
Loop Resistance Info 10-283
Export DC SPICE Subcircuit 10-284
RL Table 10-284
Path Resistance Matrix 10-285
Export DC Power Tree (General Mode Only) 10-286
Plotting DC IR Field Overlays 10-289
Overlaying DC Temperature Fields (General Mode Only) 10-292
Adding an SIwave-PSI Solution Setup 10-295
Activating the SIwave-PSI Solution Beta Feature 10-307
Using a Python Script to Configure an Ansys SIwave™ Solution Setup 10-309
Selecting SIwave Solution Setup Options 10-324
General Tab 10-324
Advanced Tab 10-326
DC Tab 10-327
DC Advanced Tab 10-329
S-Parameters Tab 10-331
Temperature Tab 10-332
Running EMI Scanner in HFSS 3D Layout (General Mode Only) 10-333
Setting Rules in EMI Scanner 10-336
Applying EMI Scanner General Settings 10-337
Applying Signal Reference Rules 10-338
Applying Wiring/Crosstalk Rules 10-348
Applying Decoupling Rules 10-357

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

Applying Placement Rules 10-367
Applying Net Integrity Rules 10-370
Applying Via Integrity Rules 10-382
Applying Power Integrity Rules 10-387
Using Tags in EMI Scanner 10-397
Assigning Tags Automatically 10-398
Assigning Tags Manually 10-401
Working with Custom Tags 10-404
Adding Custom Tags 10-404
Renaming a Custom Tag 10-406
Deleting a Custom Tag 10-407
Creating Tagging Macros 10-408
Analyzing EMI Scanner Results Using iQ-Harmony 10-410
Spectra Configuration 10-415
Creating a New Spectrum 10-419
Viewing EMI Scanner Results 10-420
Exporting EMI Scanner Results 10-423
Nexxim Solutions in HFSS 3D Layout (General Mode Only) 10-426
Add LNA Solution Setup 10-426
Setting Up a Transient Analysis in HFSS 3D Layout 10-429
AMI Analysis in HFSS 3D Layout 10-433
Eye Sources and Probes in HFSS 3D Layout 10-440
QuickEye and VerifEye Setup in HFSS 3D Layout 10-440
Adding QuickEye Setup in HFSS 3D Layout 10-441
Nexxim Solution Options for QuickEye 10-442
Quick Eye Analysis Outputs 10-444
Run the Quick Eye Analysis and Display Results 10-444
Adding VerifEye Setup in HFSS 3D Layout 10-445

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

VerifEye Analysis Outputs 10-445
Run the VerifEye Analysis and Display Results 10-446
Nexxim Solution Options for VerifEye 10-446
Subcircuit Optimization in HFSS 3D Layout and Nexxim 10-448
Adding a Frequency Sweep (HFSS 3D Layout) 10-448
Copy, Paste, and Edit Frequency Sweeps 10-453
Selecting the Sweep Type 10-454
Options for Broadband Fast Sweeps 10-455
Edit Frequency Sweep Tab 10-455
Options Tab 10-457
Choosing Frequencies for Full-Wave SPICE 10-458
Disabling or Enabling a Frequency Sweep 10-459
Q3D-Based Electrothermal Workflow in HFSS 3D Layout (Harmonic Loss) 10-460
Background 10-460
Workflow using Q3D Extractor in HFSS 3D Layout 10-461
Sanitize Layout 10-465
Clean Up Solutions 10-467
Adding a Technology to a Design (IC Mode Only) 10-468
Viewing the Technology Library 10-473
Applying Encrypted Technology Files 10-475
Modifying a Solution Setup in HFSS 3D Layout 10-477
Disabling an Analysis Setup 10-478
High Performance Computing 10-478
Remote Analysis 10-478
Prerequisites for Remote and Distributed Analysis 10-479
Configuring the Local Machine to Solve Remotely 10-480
Remote Analysis Options 10-480
Running Remote Analysis 10-481

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

Troubleshooting 10-481
Remote Solve Node = Windows 10-484
Remote Solve Node = Linux 10-485
Distributed Analysis 10-486
Beta Feature: Parallel Component Mesh Adapt for 3D Component Array 10-486
Configure a Distributed Analysis 10-487
Distributed Analysis Configuration 10-487
Distributed Analysis Configuration - Machines Tab 10-488
Manually Adding a Machine 10-489
Importing a Machine List 10-489
Machine List Details 10-490
Testing Machines 10-491
Distributed Analysis Configuration - Job Distribution Tab 10-491
Distribution Levels 10-495
Single-level Distributions 10-495
Two-Level Distributions 10-496
Activating the Q3D Distribution List Options Beta Feature 10-497
Distributed Analysis Configuration - Options Tab 10-502
Relation to Batchoptions 10-502
Adding Configurations or Accepting Edits 10-502
Selecting Optimal Configurations for Distributed Analysis 10-502
Two-Level Distribution Guidelines 10-503
Large Scale DSO for Parametric Analysis 10-504
Prerequisites for Large Scale DSO 10-506
Job Management Interface for Large Scale DSO 10-507
Large Scale DSO Command Line Syntax 10-514
Large Scale DSO Job Outputs 10-518
Output Location and Organization 10-518

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 Contents 10-518
Postprocessing Large Scale DSO Dataset Solutions 10-519
Importing a Large Scale DSO Solution 10-519
Viewing Imported Solutions 10-520
Creating a Dataset Report 10-521
Cloning a Dataset Solution 10-521
Large Scale DSO Job Monitoring 10-521
Additional Resources for Large Scale DSO Monitoring 10-522
Large Scale DSO Deployment/Configuration 10-522
Large Scale DSO Tutorial Example 10-524
Prepare the Model for Large Scale DSO Analysis 10-524
Submit the Large Scale DSO Job: Examples 10-529
Large Scale DSO Example: Post Process the Results 10-531
Large Scale DSO Known Issues/Troubleshooting 10-533
Node Order 10-533
Parallel Task Limitation for LS-DSO Parametric Variations 10-533
Job Restart 10-534
Linux-Only Issues 10-534
Interactive Scheduler Jobs 10-535
Specifying Options for Interactive Scheduler Jobs 10-535
Batchoptions for Interactive Scheduler Jobs 10-536
DSO Configuration for Interactive Scheduler Jobs 10-537
Analysis Configuration - Machines Tab 10-538
Analysis Configuration - Job Distribution Tab 10-539
Analysis Configuration - Options Tab 10-540
Design Type Options for Interactive Scheduler Jobs 10-541
Distribution Command Line Options 10-542
High Performance Computing (HPC) Integration 10-544

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

Scheduler Terminology 10-544
What a Scheduler Does 10-545
Configuring Electronics Installation for HPC 10-546
Firewall Configuration 10-547
Installation Directory Examples 10-547
Integration with Microsoft Windows® HPC Scheduler 10-547
Submitting and Monitoring Ansys EM HPC Jobs 10-548
Submitting and Monitoring Jobs for Windows HPC 10-550
Windows® HPC Job Templates 10-568
Selecting Computation Resource Units (Job Unit Type) 10-568
Windows® HPC Job Credentials 10-569
Integration with Grid Engine (GE) 10-570
GE Job Management 10-570
Installation of Ansys EM Tools on GE 10-571
GE Commands for Information about Jobs and Cluster Configuration 10-573
Monitoring GE Serial and Parallel Batch Jobs 10-576
Ansys EM Desktop -monitor Command Line Option for SGE 10-577
Example SGE qsub Command Lines 10-577
Issue with qrsh (SGE) 10-578
Issue with MainWin Core Services for SGE 10-579
Integration with Platform's Load Sharing Facility (LSF) 10-580
LSF Job Management 10-581
LSF Job Submission Guidelines 10-581
Integration of Ansys EM Products with LSF 10-582
Installing Ansys EM Tools on LSF Cluster 10-582
Using the bsub Command to Submit Batch Jobs 10-585
bsub Arguments 10-585
Example LSF bsub Command Lines (Linux Only) 10-586

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 10-586
Serial Job Requiring a Minimum of 4GB 10-586
Multi-processing Job using 4 Cores 10-586
Distributed Processing Job using 4 Engines 10-587
Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing 10-587
Shell Script (~/projects/OptimTee.csh): 10-587
Monitoring LSF Batch Jobs 10-588
Terminating LSF Batch Jobs 10-588
LSF Known Issues and Workarounds 10-589
LSF Troubleshooting 10-589
Integration with PBS (Portable Batch System) 10-590
PBS Job Management 10-590
Non Standard Installations for PBS 10-591
PBS Limitations 10-591
Submitting Ansys EM PBS Batch Jobs 10-593
Monitoring PBS Batch Jobs 10-594
qsub Arguments 10-595
Example PBS qsub Command Lines 10-596
Using the Command Line to Submit HPC Jobs 10-598
Distributed Jobs 10-598
Serial Job on a Single Processor 10-599
Distributed Analysis and Multiprocessing in the Same Job 10-599
Integrating Ansys EM Tools with Third-Party Schedulers 10-599
Introduction 10-599
Serial Jobs 10-600
Parallel Jobs 10-600
Common Requirements for Running Jobs 10-601

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

Using a Shared Library (Linux) or a DLL (Microsoft Windows) 10-601
Build Information for Scheduler Proxy Library 10-602
Implementation Details for Custom Scheduler Integration 10-603
IsProductLaunchedInYourEnvironment 10-603
GetTempDirectory 10-604
GetMachineListAvailableForDistribution 10-605
GetMessageStringToRegisterForSigTerm 10-605
LaunchProcess 10-606
GetUseRsmForEngineLaunch 10-607
GetThisJobID 10-608
GetSchedulerDisplayName 10-609
Scheduler Proxy Interfaces 10-610
Testing Scheduler Integration 10-616
Testing IsProductLaunchedInYourEnvironment 10-616
Testing GetSchedulerDisplayName and GetThisJobID 10-616
Testing GetTempDirectory 10-617
Testing GetMachineListAvailableForDistribution 10-617
Testing LaunchProcess 10-617
Testing GetUseRsmForEngineLaunch 10-618
Troubleshooting Custom Scheduler Integration 10-618
None of the Proxy Functions are Called 10-618
Troubleshooting IsProductLaunchedInYourEnvironment Function 10-619
Troubleshooting GetSchedulerDisplayName 10-619
Troubleshooting GetThisJobID 10-620
Troubleshooting GetTempDirectory 10-620
Troubleshooting GetMachineListAvailableForDistribution 10-620
Troubleshooting LaunchProcess 10-621
Troubleshooting GetUseRsmForEngineLaunch 10-621

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

Using an IronPython Program for Integration with a Scheduler 10-621
GetName [IronPython] 10-623
GetDescription [IronPython] 10-623
IsProductLaunchedInYourEnvironment [IronPython] 10-624
GetSchedulerDisplayName [IronPython] 10-624
GetThisJobID [IronPython] 10-625
GetUseRsmForEngineLaunch [IronPython] 10-625
GetTempDirectory [IronPython] 10-626
GetMessageStringToRegisterForSigTerm [IronPython] 10-627
GetMachineListAvailableForDistribution [IronPython] 10-627
LaunchProcess [IronPython] 10-628
Selecting a Scheduler 10-629
Submitting a Job 10-631
Using Advanced Job Submission Options 10-636
Batchoptions 10-637
Environment Variables 10-638
Batch Extract 10-639
Customize Job Submission 10-640
Using the Command Line to Submit HPC Jobs 10-640
Distributed Jobs 10-640
Serial Job on a Single Processor 10-640
Distributed Analysis and Multiprocessing in the Same Job 10-641
Job Import and Export 10-641
Scripting 10-641
Multi-Step Job Submission 10-642
Windows to Linux Job Submission 10-647
Prerequisites for Job Submission 10-648
Prerequisites for Job Monitoring 10-649

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

Supported Schedulers 10-649
Job Submission Scripting 10-655
Monitoring Jobs 10-656
Monitor Job Window 10-656
Monitoring Ansys Cloud Burst Jobs 10-658
Monitoring Large Scale DSO Jobs 10-659
Web Client for Batch Solve Monitoring and Reporting 10-659
Integration with Ansys Remote Simulation Management (RSM) 10-665
Changing the AnsoftRSMService Listening Port 10-680
AnsoftRSMService Configuration 10-681
Ansoft Electromagnetics Desktop Configuration 10-682
Support for Open MPI on Linux [Beta] 10-684
Running HPC Diagnostics 10-688
Site-Specific Diagnostics Job 10-691
Changing Solution Priority for System Resources 10-693
Aborting an Analysis 10-694
11 - Circuit and Layout Definition Libraries 11-1
Working with Definition Libraries 11-1
System, User, and Personal Library Directories 11-2
Changing the Locations of Libraries 11-2
Library and Project Definitions 11-2
Updating Project Definitions from Library Definitions 11-3
Exporting and Importing Definition Archives 11-3
Exporting Definition Archives 11-3
Importing Definition Archives 11-6
Managing Library Files 11-8
Library Search Precedence 11-9
Component Creation Sequence 11-9

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 Component Editor 11-10
Creating and Editing Components 11-11
Editing an Existing Component 11-11
Creating a New Component 11-12
The Edit Component Window 11-12
Components General Tab 11-12
Components Miscellaneous Tab 11-14
Components Terminals Tab 11-16
Components Solver On Demand Tab 11-18
HFSS 3D Layout as Solver On Demand 11-21
Edit Component Properties Window 11-22
Parameter Statistics Display 11-23
Reserved Component Parameters 11-25
Component CosimDefinition Property 11-26
Netlist String Syntax 11-26
Using the Symbol Editor 11-29
Creating a New Symbol 11-30
Editing an Existing Symbol 11-31
Symbol Editor Operations 11-31
Symbol Draw Menu 11-32
Symbol Menu 11-38
Import File (Symbol) 11-40
Editing Pin Properties 11-41
Edit Symbol Pin Locations 11-42
Pin List 11-42
Pin Location Editor 11-44
Bus Pin Editor 11-45
Committing Changes 11-47

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

Symbol Property Display Setup 11-47
Symbol Grid Setup window 11-48
Using the Footprint Editor 11-49
Opening the Footprint Editor 11-50
Creating a New Footprint 11-50
Editing an Existing Footprint 11-51
Load a Footprint into the Current Project 11-51
Footprint Editor Operations 11-51
Footprint Edit Menu 11-52
Footprint Shortcut Menu 11-53
Footprint View Menu 11-53
Footprint Draw Menu 11-56
Defining Footprint Handles 11-60
Using Scripts to Define Footprints 11-61
Layout Host Object 11-61
LayoutHost Properties 11-61
LayoutHost Methods 11-62
LayoutHost General Methods 11-62
LayoutHost Shape Creation Methods 11-63
LayoutHost Relative Shape Placement Methods 11-66
LayoutHost Port and Pin Methods 11-67
LayoutHost Via and Padstack Methods 11-69
ElementPars Object 11-71
ElementPars Methods 11-71
Points Object 11-71
Points Object Properties 11-71
Points Object Methods 11-72
Geom Object 11-73

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

Geom Object Properties 11-73
Geom Object Methods 11-73
Edge Object 11-76
Edge Object Properties 11-76
Edge Object Methods 11-77
Via Object 11-77
Via Object Properties 11-77
Via Object Methods 11-78
Using the Padstack Editor 11-82
Creating a New Padstack 11-82
Editing a Padstack 11-82
Editing a Polygon in a Padstack 11-88
Pad Behavior in Vias and Pins 11-90
Editing Via and Pin Padstacks in Layout 11-91
Initial Padstack Definition Layers 11-92
Pin and Net Padstack Operations 11-93
Using the Estimate Module in HFSS 3D Layout 11-95
Estimating Transmission Line Quantities 11-96
Estimating Characteristic Impedance 11-96
Estimating Width 11-97
Estimating Guided Wavelength and Effective Relative Permittivity 11-97
Estimating Line Loss 11-98
Estimating Matches Between Impedances 11-99
Estimating Quarter-Wave Transformer Dimensions 11-99
Estimating Tuning Stub Parameters 11-100
Estimating Dimensions of Printed Antennas 11-102
Estimating Corners-Truncated CP Element Parameters 11-103
Estimating Nearly Square CP Element Parameters 11-104

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

Estimating Rectangular Patch Parameters 11-104
Using the Microwave Calculator 11-106
Converting between Microwave Parameters 11-106
Converting Between dB and Ratio 11-107
Backdrilling 11-108
Viewing Backdrill Properties 11-109
Defining Backdrill Parameters 11-109
Viewing Backdrill Parameters 11-110
Activating/Deactivating Backdrill Subgroup Boxes 11-113
Setting Parameters in the Backdrill Definition Window 11-114
Using Project Variables to Modify a Backdrill 11-119
Using the Models Editor 11-123
Using the Material Editor 11-124
The View / Edit Material Window 11-124
Editing an Existing Material 11-127
Creating a New Material 11-128
Using Vendor Components 11-128
Using RF Vendor Library Components 11-128
Encrypted Libraries 11-129
Creating and Managing Encrypted Libraries 11-130
Password Assignment Types 11-130
Setting Up Passwords and Encrypted Libraries 11-132
Generate a New Password Protected Resource 11-132
Using the Library Editor 11-137
Starting the Library Editor 11-137
Edit Libraries Window 11-138
Exporting Hierarchical Components 11-140
12 - Optimetrics 12-1

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

Parametric Overview 12-2
Setting Up a Parametric Analysis 12-3
Adding a Variable Sweep Definition 12-4
Specifying Variable Values for a Sweep Definition 12-5
Synchronizing Variable Sweep Definitions 12-6
Modifying a Variable Sweep Definition Manually 12-6
Overriding a Variable's Current Value in a Parametric Setup 12-7
Specifying a Solution Setup for a Parametric Setup 12-8
Specifying the Solution Quantity to Evaluate for Parametric Analysis 12-8
Setting up Calculations for Optimetrics 12-9
Specifying a Solution Quantity's Calculation Range 12-10
Viewing Results for Parametric Solution Quantities 12-11
Using Distributed Analysis 12-11
Adding a Parametric Sweep from a File 12-12
Validating a Parametric Setup 12-13
Optimization Overview 12-16
Choosing an Optimizer 12-16
Legacy Optimizers 12-18
Quasi Newton (Gradient) 12-18
Pattern Search (Search-Based) 12-22
Sequential Nonlinear Programming (Gradient) 12-24
Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) 12-25
Genetic Algorithm (Random Search) 12-26
MATLAB Optimizer 12-27
Multi-Objective Genetic Algorithm (MOGA) 12-32
MOGA Workflow 12-33
MOGA Steps 12-34
MOGA Steps to Generate a New Population 12-35

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

Convergence Rate % and Initial Finite Difference Delta % in NLPQ and MISQP .. 12-38
Mixed-Integer Sequential Quadratic Programming 12-40
Adaptive Multiple Objective Optimization 12-41
AMO Workflow 12-41
AMO Steps 12-42
Adaptive Single Objective Optimization 12-44
ASO Workflow 12-44
AMO Steps 12-46
Optimization Variables and the Design Space 12-47
Setting Up an Optimization Analysis 12-48
Setting Up the Quasi Newton (Gradient) Optimizer 12-49
Setting Up the Pattern Search (Search-based) Optimizer 12-51
Merit-Based Sequential Quadratic Programming(Gradient) Optimizer Setup 12-54
Setting Up the Sequential Nonlinear Programming (Gradient) Optimizer 12-55
Setting Up the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer 12-57
Setting Up the Genetic Algorithm (Random Search) Optimizer 12-59
Setting Up the MATLAB Optimizer 12-61
Screening (Search Based) Optimizer Setup 12-62
Setting Up the Multi-Objective Genetic Algorithm(Random Search) Optimizer 12-65
Setting Up the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer 12-71
Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) 12-76
Setting Up the Multi-Objective Genetic Algorithm(Random Search) Optimizer 12-81
Adaptive Multiple-Objective (Random Search) Optimizer Setup 12-87
Adaptive Single-Objective (Gradient) Optimizer Setup 12-92
Setting the Maximum Iterations for an Optimization Analysis 12-97
Cost Function 12-98

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 12-99
Cost Function Noise 12-99
Adding a Cost Function 12-99
Specifying a Solution Quantity for a Cost Function Goal 12-102
Setting the Calculation Range of a Cost Function Goal 12-102
Setting a Goal Value 12-103
Specifying a Single Goal Value 12-103
Specifying an Expression as a Goal Value 12-104
Specifying a Variable-Dependent Goal Value 12-104
Goal Weight 12-105
Modifying the Starting Variable Value for Optimization 12-106
Setting the Min. and Max. Variable Values for Optimization 12-107
Edit Calculation Range 12-107
Overriding the Min. and Max. Variable Values for a Single Optimization Setup 12-110
Changing the Min. and Max. Variable Values for Every Optimization Setup 12-111
Step Size 12-111
Setting the Min. and Max. Step Sizes 12-112
Setting the Min and Max Focus 12-113
Equalizing the Influence of Different Optimization Variables 12-113
Setting the Min and Max Focus Values 12-114
Solving a Parametric Setup Before an Optimization 12-114
Solving a Parametric Setup During an Optimization 12-114
Automatically Updating a Variable's Value after Optimization 12-115
Changing the Cost Function Norm 12-115
Explanation of L1, L2, and Max Norms in Optimization 12-115
Example of a More Complex Cost Function 12-118
Advanced Genetic Algorithm Optimizer Options 12-119
Sensitivity Analysis Overview 12-122

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 Primary Output 12-122
Setting Up a Sensitivity Analysis 12-122
Setting the Maximum Iterations Per Variable 12-124
Setting Up an Output Parameter 12-125
Specifying a Solution Quantity for an Output Parameter 12-125
Setting the Calculation Range of an Output Parameter 12-127
Modifying the Starting Variable Value for Sensitivity Analysis 12-127
Setting the Min. and Max. Variable Values 12-128
Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup 12-128
Changing the Min. and Max. Variable Values for Every Sensitivity Setup 12-129
Setting the Initial Displacement 12-129
Solving a Parametric Setup before a Sensitivity Analysis 12-129
Solving a Parametric Setup during a Sensitivity Analysis 12-130
Performing Worst Case Analysis 12-130
Statistical Analysis Overview 12-134
Setting Up a Statistical Analysis 12-135
Setting the Maximum Iterations for a Statistical Analysis 12-136
Specifying the Solution Quantity to Evaluate for Statistical Analysis 12-137
Setting the Solution Quantity's Calculation Range 12-138
Setting the Distribution Criteria 12-139
Overriding the Distribution Criteria for a Single Statistical Setup 12-139
Changing the Distribution Criteria for Every Statistical Setup 12-140
Statistical Cutoff Probability 12-141
Edit Distribution 12-142
Modifying the Starting Variable Value for Statistical Analysis 12-143
Solving a Parametric Setup During a Statistical Analysis 12-144
Using the Fast Calculation-Update Algorithm 12-144
Tuning Overview 12-147

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 Variable 12-148
Applying a Tuned State to a Design 12-150
Saving a Tuned State 12-151
Reverting to a Saved Tuned State 12-151
Resetting Variable Values after Tuning 12-152
Copying Meshes in Optimetrics Sweeps 12-152
Adding an Expression in the Output Variables Window 12-153
Excluding a Variable from an Optimetrics Analysis 12-153
Modifying the Value of a Fixed Variable 12-154
Linear Constraints 12-154
Setting a Linear Constraint 12-155
Modifying a Linear Constraint 12-156
Deleting a Linear Constraint 12-156
Running an Optimetrics Analysis 12-157
Viewing Analysis Results for Optimetrics Solutions 12-157
Viewing Solution Data for an Optimetrics Design Variation 12-158
Viewing an Optimetrics Solution's Profile Data 12-158
Viewing Results for Parametric Solution Quantities 12-159
Plotting Solution Quantity Results vs. a Swept Variable 12-159
Viewing Cost Results for an Optimization Analysis 12-160
Plotting Cost Results for an Optimization Analysis 12-161
Viewing Output Parameter Results for a Sensitivity Analysis 12-161
Plotting Output Parameter Results for a Sensitivity Analysis 12-161
Viewing Distribution Results for a Statistical Analysis 12-162
Plotting Distribution Results for a Statistical Analysis 12-163
Link to DesignXplorer 12-163
13 - Modifying the 3D Model View 13-1
Change the View 13-1

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

Show or Hide Design Objects ... 13-2
Change Object Visualization ... 13-2
Change the Background ... 13-2
View Options: 3D UI Options ... 13-2
Panning the View ... 13-8
Zooming In and Out ... 13-9
Zoom In and Zoom Out Commands ... 13-11
Zoom Previous Command ... 13-12
Zooming into a Rectangular Area ... 13-14
Zooming in or out Using a Mouse Wheel ... 13-15
Zoom to a Selected Boundary or Excitation ... 13-15
Modifying the View Orientation ... 13-16
Apply an Orientation to the Current View ... 13-17
Rotating or Spinning the View ... 13-18
Changing the Model View with Alt+Double-Click Areas ... 13-20
Fitting Objects in the View Window ... 13-20
Fitting All Objects in a View Window ... 13-21
Hiding Objects from View ... 13-21
Active View Visibility Dialog Box ... 13-23
Rendering Objects as Wireframes or Solids ... 13-24
Setting the Default View Rendering Mode ... 13-25
Enabling Optimized Rendering (Beta) ... 13-25
Setting the Enhanced Display of Material Color and Transparency ... 13-26
Setting the Curved Surface Visualization ... 13-28
Using Clip Planes ... 13-29
Modifying the Lighting ... 13-33
Setting the Projection View ... 13-34
Setting the Background Color ... 13-35

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 13-38
Showing or Hiding the Axes 13-38
Show the Axes for Selected Objects 13-38
Enlarging or Shrinking the Axes 13-39
Showing or Hiding the Triad Axes 13-39
Choosing Grid Settings 13-39
Setting the Grid Type 13-41
Setting the Grid Style 13-42
Setting the Grid Density and Spacing 13-42
Setting the Grid's Visibility 13-44
Setting the Grid Plane 13-45
14 - Postprocessing and Generating Reports 14-1
Fields Post Processing in HFSS 3D Layout 14-2
Viewing Solution Data 14-3
Viewing a Solution Profile 14-4
Viewing Convergence Data 14-7
Viewing the Number of Completed Passes 14-10
Viewing the Max Magnitude of Delta S Between Passes 14-10
Viewing the Delta Magnitude Energy 14-11
Plotting Convergence Data 14-11
Viewing Matrix Data 14-12
Selecting the Matrix Display Format 14-16
Exporting Matrix Data 14-18
Renaming Matrix Data 14-21
Reordering Matrix Data 14-22
Exporting Equivalent Circuit Data 14-24
Exporting W-element Data 14-27
Viewing Mesh Statistics 14-30

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 14-32
Creating Reports 14-33
Creating a New Report 14-34
Context Section for Reports 14-38
Using Families Tab for Reports 14-38
Filtering Quantity Selections for the Reporter 14-41
Deleting Reports 14-42
Modifying Reports 14-42
Modify Report: Selecting Use All Values or Making Selection 14-46
Modify Report: Using the Edit Sweep Dialog Box 14-47
Generating Reports For Characteristic Mode Analysis 14-48
Characteristic Mode Data Report 14-53
Modal Significance vs Frequency 14-56
Characteristic Value vs Frequency 14-57
Characteristic Angle vs Frequency 14-58
Modal Coefficient Plot 14-59
Generating Reports For Monostatic RCS 14-59
Monostatic RCS Reports 14-62
Creating a Report from an Ansoft Report Data File 14-64
Zooming and Fitting Reports 14-64
Zooming In and Out 14-64
Zoom In and Zoom Out Commands 14-66
Zoom Previous Command 14-66
Modifying the Background Properties of a Report 14-68
Modifying the Legend in a Report 14-71
Exporting Ansoft Report Data Format Files 14-74
Exporting Reports as Graphics 14-76
Modifying the Background Properties of a Report 14-78

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 Legend in a Report 14-80
Creating Custom Report Templates and Defaults 14-83
Exporting Ansoft Report Data Format Files 14-84
Exporting Reports as Graphics 14-86
Report File Formats 14-88
Creating Custom Report Templates and Defaults 14-89
Selecting the Report Type 14-90
Selecting the Display Type 14-92
Creating 2D Rectangular Plots 14-93
Creating 2D Rectangular Stacked Plots 14-95
Creating 3D Rectangular Plots 14-97
Creating Rectangular Contour Plots 14-104
Creating Sine Space Plots 14-107
Sine Space Plots 14-107
Creating 2D Polar Plots 14-111
Reviewing 2D Polar Plots 14-112
Creating 3D Polar Plots 14-113
Creating Smith Charts 14-122
Creating Smith Contour Charts 14-123
Creating Data Tables 14-125
Creating Radiation Patterns 14-127
Delta Markers in 2D Reports 14-129
Plotting in the Time Domain 14-129
TDR Windowing Functions 14-133
Working with Traces 14-136
Editing Trace Properties 14-137
Editing the Display Properties of Traces 14-138
Adding Data Markers to Traces 14-141

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

Y Markers in Stacked XY Plots 14-144
Discarding Report Values Below a Specified Threshold 14-151
Adding Characteristics to a Trace 14-152
Adding a Recently Used Trace Characteristic 14-152
Adding a Trace Characteristic from Favorites 14-152
Adding Trace Characteristics to your Favorites 14-153
Adding Characteristics Using the Add Trace Characteristics Window 14-155
Removing All Trace Characteristics 14-157
Removing Traces 14-157
Copy and Paste of Report and Trace Definitions 14-157
Copy and Paste of Report and Trace Data 14-158
Limit Lines in Cartesian Plots 14-159
Sweeping a Variable in a Report 14-167
Sweeping Values Across a Distance 14-167
Sweeping Values Across a Sphere 14-168
Selecting a Function for a Plot 14-169
Selecting Solution Quantities to Plot 14-175
Selecting a Field Quantity to Plot 14-181
Selecting a Far-Field Quantity to Plot 14-182
Prerequisites 14-182
Set Up a Far Field Infinite Sphere 14-190
Plotting Vertical Cross-Sections of Far Fields 14-196
Plotting Horizontal Cross-Sections of Far Fields 14-197
Selecting a Near-Field Quantity to Plot 14-198
Prerequisites 14-198
To select a near-field quantity to plot: 14-198
Overlaying Near Field Reports 14-203
Selecting Antenna Parameters to Plot 14-205

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

Antenna Parameters 14-210
Selecting an Emission Test Quantity to Plot 14-210
Plotting Imported Solution Data 14-211
Setting a Range Function 14-211
Range Functions 14-212
Eye Measurement Range Function Parameters 14-217
Perform FFT on a Report 14-217
FFT Window Functions 14-218
Apply FFT to Report Functions 14-220
Perform TDR on Report 14-222
Window Functions and Time Domain Plotting 14-223
Animated Reports 14-236
Overlaying Surface Currents on a 3D View 14-236
Overlaying Far Fields on a 3D View 14-239
Overlaying Near Fields on a 3D View 14-241
Frequency Animation 14-244
Phase Animation 14-247
Changing the Design Point 14-251
Specifying Output Variables 14-252
Selecting Solution Quantities to Plot 14-255
Function List for Output Variables 14-261
Plotting Field Overlays in HFSS 3D Layout 14-262
Creating Field Overlays 14-262
Field Quantities 14-267
Creating Animations 14-269
Creating Phase Animations 14-271
Creating Frequency Animations 14-274
Creating Geometry Animations 14-277

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

Creating 2D Radiation Field Plot Overlay Animations 14-280
Controlling the Animation's Display 14-281
Exporting Animations 14-283
Exporting Ensight from the Select Animation Dialog Box 14-283
Exporting from the Animation Control Dialog Box 14-285
Using the Fields Calculator 14-287
Scripting Support 14-289
Opening the Fields Calculator 14-289
Context Area 14-290
The Calculator Stack 14-290
Stack Commands 14-292
Pop 14-292
RlUp 14-292
RlDn 14-292
Exch 14-292
Clear 14-292
Undo 14-293
Registers 14-293
Enlarging the Register Display Area 14-295
Units of Measure 14-296
Input Commands 14-296
Quantity Command 14-297
Geometry Command 14-298
Constant Command 14-301
Number Command 14-303
Function Command 14-303
Geom Settings Command 14-305
Read Command 14-306

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

Output Vars [Input for Eigenmode problems] 14-306
General Commands 14-306
Steps for Calculating the Complex Vector Electric Field 14-310
Scalar Commands 14-310
Vec? Command 14-312
Pow Command 14-312
Square Root Command 14-313
Trigonometry Commands 14-313
d/d? (Partial Derivative) Command 14-313
Integral Command 14-313
Min Command 14-314
Max Command 14-314
Gradient Command 14-315
Ln Command 14-315
Log Command 14-315
Std Command 14-315
Vector Commands 14-315
Scal? Command 14-317
Matl Command 14-317
Mag Command 14-318
Dot Command 14-318
Cross Command 14-318
Divg Command 14-318
Curl Command 14-318
Tangent Command 14-318
Normal Command 14-319
Unit Vec Command 14-320
X Form 14-320

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 14-321
Value Command 14-322
Eval Command 14-324
Write Command 14-325
Export Command 14-326
Calculating Derived Field Quantities Using Named Expressions 14-331
List of Field Quantities in a Q3D Solution Setup 14-333
DC R/L Fields 14-333
AC R/L Fields 14-333
CG Fields 14-334
DC R/L Thin Conductor Fields 14-334
Named Expression Library 14-334
Exiting the Fields Calculator 14-335
Fields Calculator Recipes 14-336
Calculating Numerical Quantities 14-336
Calculating Quantities for 2D (Line) Plot Outputs 14-336
Calculating Quantities for 3D (Surface or Vector) Plot Outputs 14-336
Calculating Quantities for 3D (Volume) Plot Outputs 14-336
Calculating Quantities for Animated Outputs 14-336
Creating User Defined Named Expressions Library 14-336
Recipe Format 14-337
EXAMPLE: Title of Current Calculation 14-337
Calculating the Current along a Wire or Trace 14-337
Calculating the Voltage Drop along a Line 14-338
Calculating Net Power Flow through a Surface 14-340
Calculating the Average of a Field Quantity on a Surface 14-341
Calculating the Peak Electrical Energy in a Volume 14-342
Calculating the Q of a Resonant Cavity 14-343

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

Calculating the Max Value of a Field Quantity in a Volume 14-346
Calculating Dielectric (or Conduction) Loss in a Volume 14-347
Plotting Wave Impedance along a Line 14-347
Plotting the Phase of E Tangential to a Line/Curve 14-349
Plotting the Maximum Magnitude of E Tangential to a Line/Curve 14-350
Plotting the E-Field Magnitude Normal to a Surface 14-351
Generating an Iso-Surface Contour for a Given Field Value 14-352
Generating an Animation on Multiple Planes with a Positional Variable 14-353
Generating Cartesian/Cylindrical/Spherical-Coordinate Field-Components Library 14-354
User-Defined Outputs (UDOs) 14-356
Named Probes and Properties in User Defined Outputs 14-356
14-361
Computation of Traces Based UDO Calculations 14-361
Dimensions Reduction by UDO Calculations 14-362
Dynamic Probes 14-362
User-Defined Outputs: Python Script API 14-362
UDO Extension Implementation 14-363
Import Statements 14-363
UDOExtension Class 14-363
Validate 14-364
IUDOPluginExtension Abstract Class 14-365
GetUDSName 14-365
GetUDSDescription 14-366
GetUDSSweepNames 14-366
GetCategoryNames 14-366
GetQuantityNames 14-367
GetQuantityInfo 14-368

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

GetInputUDSParams 14-368
GetDynamicProbes 14-371
Compute 14-372
Data Types Used in UDO Python Scripts 14-374
Constants 14-374
Abstract Classes 14-375
IUDSInputData 14-375
GetDoubleProbeData 14-375
GetSweepsDataForProbe 14-376
GetComplexProbeData 14-376
GetSweepNamesForProbe 14-377
GetRequiredQuantities 14-378
GetVariableValues 14-378
GetInterpolationOrdersData 14-379
IUDSOutputData 14-379
SetSweepsData 14-380
SetDoubleQuantityData 14-380
SetComplexQuantityData 14-381
Working With Properties for UDO 14-381
IPropertyList Abstract Class 14-382
IProperty Abstract Class 14-382
INumberProperty Abstract Class 14-382
ITextProperty Abstract Class 14-383
IMenuProperty Abstract Class 14-383
Other Application-Specific Classes Used in Python Scripts 14-384
Constants Class 14-384
UDSProbeParams Class 14-384
UDSDynamicProbes Class 14-385

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

QuantityInfo Class 14-385
IProgressMonitor Abstract Class 14-386
SetTaskName 14-387
SetSubTaskName 14-387
BeginTask 14-388
SetTaskProgressPercentage 14-388
CheckForAbort 14-388
EndTask 14-389
Using .NET Collection Classes and Interfaces in Python Scripts 14-389
User-Defined Outputs: Messaging Methods 14-391
AddErrorMessage 14-396
AddInfoMessage 14-397
AddWarningMessage 14-397
User-Defined Outputs: Script Organization 14-397
Using Script Libraries 14-398
Using Additional .NET Assemblies 14-398
User-Defined Documents (UDDs) 14-399
Create User-Defined Document Inputs 14-400
UDD Document Creation and Display 14-402
Managing Documents Listed in the Project Window Under Results 14-403
Documents Folder's Context Menu 14-404
Document Folder's Property Window 14-404
Viewing UDDs with an HTML Web Browser 14-405
UDD Script Libraries 14-406
User-Defined Documents: Python Script API 14-407
Import Statements 14-407
Data Types Used in UDD Python Scripts 14-408
Constants 14-408

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 14-408
IProgressMonitor Abstract Class 14-408
SetTaskName 14-409
SetSubTaskName 14-409
BeginTask 14-410
SetTaskProgressPercentage 14-410
CheckForAbort 14-411
EndTask 14-411
IUDDPluginExtension Abstract Class 14-412
GetUDDName 14-412
GetUDDDescription 14-413
ShowDefaultSetupDialog 14-413
GetUDDInputParams 14-413
Generate 14-415
SetupUDDInputParams 14-417
HandleUDDEvents 14-418
GetUDDSchema 14-419
GetUDDStyleSheetForHtml 14-419
GetUDDStyleSheetForPdf 14-420
GetFopExecutable 14-420
GetUDDAppContext 14-421
GetUDDDesignContext 14-421
UDDExtension Class 14-422
UDDInputData Class 14-422
UDDInputBool 14-423
UDDInputDouble 14-423
UDDInputSolution 14-423
UDDInputText 14-424

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

UDDInputTrace 14-424
UDDInputParams Class 14-425
UDD Input Interfaces 14-426
Document Generator Interfaces 14-427
Post-processing and Generating Reports for 2D and Circuit 14-432
Creating Reports for 2D and Circuit Projects 14-432
2D and Circuit Report Types 14-433
2D and Circuit New Report Window 14-434
2D and Circuit Display Types 14-438
2D and Circuit Rectangular Plot 14-439
2D and Circuit Polar Plot 14-441
2D and Circuit Data Table 14-442
2D and Circuit Smith Chart 14-443
2D and Circuit Rectangular Stacked Plot 14-444
2D and Circuit 3D Rectangular Plot 14-445
2D and Circuit 3D Polar Plot 14-446
2D and Circuit Rectangular Contour Plot 14-447
2D and Circuit Smith Contour Plot 14-448
Circuit Eye Diagram Plot 14-448
2D and Circuit Stacked Eye Diagram Plot 14-449
Circuit Statistical Eye Plot 14-451
2D and Circuit Plot-On-Schematic 14-454
2D and Circuit Reports with Differential Pairs 14-456
2D and Circuit Creating a Report from a File 14-457
Setting 2D and Circuit Report Options 14-458
Setting Report Setup Options (2D and Circuit) 14-458
Setting Report2D Options 14-459
Report2D Options: Curve Tab 14-459

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 Tab 14-459
Report2D Options: Grid Tab 14-460
Report2D Options: Header Tab 14-460
Report2D Options: Note Tab 14-460
Report2D Options: Legend Tab 14-461
Report2D Options: Marker Tab 14-461
Report2D Options: Marker Table Tab 14-462
Report2D Options: X/Y Markers Tab 14-462
Report2D Options: Digital Tab 14-464
Report2D Options: General Tab 14-464
Report2D Options: Table Tab 14-465
Modifying 2D and Circuit Reports 14-465
Modifying 2D and Circuit Report Data 14-465
Modifying the Background Properties of a 2D or Circuit Report 14-466
Modifying the Legend in a 2D or Circuit Report 14-468
Plotting Circuit Spectral Domain Data 14-469
Delta Markers in 2D Reports (2D and Circuit) 14-472
Limit Lines in 2D and Circuit Cartesian Plots 14-472
2D and Circuit Spinning a 3D Report 14-475
Reviewing 2D Polar Plots 14-475
2D Overlaying Surface Currents on a 3D View 14-476
2D Overlaying Far Fields on a 3D View 14-478
2D Overlaying Near Fields on a 3D View 14-480
2D and Circuit Updating Post-processing Data 14-484
Deleting 2D and Circuit Reports 14-486
Working with Traces in 2D and Circuit Reports 14-486
Adding a Trace to a 2D or Circuit Report 14-487
Editing 2D and Circuit Trace Properties 14-488

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 (2D and Circuit) 14-489
Adding Data Markers to 2D and Circuit Traces 14-490
Discarding 2D and Circuit Report Values below a Specified Threshold 14-491
Add 2D or Circuit Trace Characteristics 14-492
Removing Traces from 2D and Circuit Reports 14-493
Define Traces Using Range Functions (2D and Circuit) 14-493
Copy and Paste of 2D and Circuit Report and Trace Definitions 14-505
Copy and Paste 2D and Circuit Report and Trace Data 14-506
Variables, Quantities, and Functions in 2D and Circuit Reports 14-506
Sweeping a Variable in a 2D or Circuit Report 14-506
2D and Circuit Selecting a Function 14-507
Selecting Solution Quantities to Plot (2D and Circuit) 14-512
Selecting a Field Quantity to Plot (2D and Circuit) 14-513
2D and Circuit Selecting a Far-Field Quantity to Plot 14-514
2D and Circuit Directive Gain 14-515
2D and Circuit Radiation Efficiency 14-515
Setting a Range Function (2D and Circuit) 14-516
Handling a Large Number of 2D or Circuit Ports 14-517
Working with Eye Diagrams 14-518
Modifying Circuit Eye Diagram Reports 14-518
Eye Measurements (2D and Circuit) 14-521
Eye Measurement Range Function Parameters (2D and Circuit) 14-527
Using Stacked Eye Diagrams 14-527
2D and Circuit PAM-4 Eye Plot 14-530
Statistical Eye Plots of PAM-4 Results 14-531
Voltage Plots of PAM-4 Results 14-532
Eye Measurements for PAM-4 Plots 14-534
Specifying Output Variables in 2D and Circuit Reports 14-536

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

2D and Circuit Adding a New Output Variable 14-536
2D and Circuit Building an Expression Using Existing Quantities 14-536
Deleting Output Variables (2D and Circuit) 14-538
Viewing Matrix Data in 2D and Circuit Reports 14-538
2D and Circuit Selecting the Matrix Display Format 14-539
Renaming 2D and Circuit Matrix Data 14-540
Animation in 2D and Circuit Reports 14-540
2D Frequency Animation 14-540
2D Phase Animation 14-543
Changing the 2D Design Point 14-547
User-Defined Outputs in 2D and Circuit Reports: Introduction 14-548
Named Probes and Properties in user-defined Outputs (2D and Circuit) 14-549
Computation of Traces Based UDO Calculations (2D and Circuit) 14-550
Dimensions Reduction by UDO Calculations (2D and Circuit) 14-550
Dynamic Probes (2D and Circuit) 14-550
user-defined Outputs: Python Script API (2D and Circuit) 14-551
UDO Extension Implementation (2D and Circuit) 14-551
Import Statements (2D and Circuit) 14-551
Optional Functions in IDO Extension Abstract Class (2D and Circuit) 14-552
Data Types Used in Python Script (2D and Circuit) 14-552
Working With Properties for UDO (2d and Circuit) 14-552
Other Application Specific Classes Used in Python Scripts (2D and Circuit) 14-552
Other Application Specific Classes Used in Python Scripts (2D and Circuit) 14-553
Constants Class (2D and Circuit) 14-553
Other Application Specific Classes Used in Python Scripts (2D and Circuit) 14-553
Constants Class (2D and Circuit) 14-553
Other Application Specific Classes Used in Python Scripts (2D and Circuit) 14-554
Constants Class (2D and Circuit) 14-554

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 (2D and Circuit) 14-555
Constants Class (2D and Circuit) 14-555
Other Application Specific Classes Used in Python Scripts (2D and Circuit) 14-555
Constants Class (2D and Circuit) 14-555
Using .NET Collection Classes and Interfaces in Python Scripts (2D and Circuit) 14-556
user-defined Outputs: Messaging Methods (2D and Circuit) 14-558
user-defined Outputs: Script Organization (2D and Circuit) 14-564
Using Script Libraries (2D and Circuit) 14-565
Using Additional .NET Assemblies (2D and Circuit) 14-565
user-defined Documents (UDDs) (2D and Circuit) 14-566
Create user-defined Document window Inputs (2D and Circuit) 14-567
UDD Document Creation and Display (2D and Circuit) 14-569
Managing Documents Listed in the Project Manager window Under Results (2D and Circuit) 14-570
Viewing UDDs with an HTML Web Browser (2D and Circuit) 14-572
UDD Script Libraries (2D and Circuit) 14-573
user-defined Definitions: Python Script API (2D and Circuit) 14-574
Import Statements (2D and Circuit) 14-574
UDDExtension Class (2D and Circuit) 14-575
IUDDPluginExtension Abstract Class (2D and Circuit) 14-575
Data Types Used in Python Script (2D and Circuit) 14-582
Constants Class (2D and Circuit) 14-582
UDDInputParams Class (2D and Circuit) 14-582
IProgressMonitor Abstract Class (2D and Circuit) 14-583
UDD Input Interfaces (2D and Circuit) 14-584
user-defined Document Scripting Interface (2D and Circuit) 14-587
The UserDefinedDocument Data Format (2D and Circuit) 14-588

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

Document Generator Interfaces (2D and Circuit) 14-590
Technical Notes for 2D and Circuit Reports 14-596
Unit Interval and Amplitude Values for Circuit Bathtub Curves 14-597
Apply Receiver Jitter and Noise Controls Simultaneously (2D and Circuit) 14-601
EM Subdesign Plotting on the Top-Level Design (2D and Circuit) 14-604
Post Processing for Transient in Layout 14-606
Chip Model Analyzer (CMA) 14-612
PinToPin Utility 14-612
Launching PinToPin Utility 14-613
Importing a File 14-614
Selecting a Simulation Mode 14-616
Exporting a File 14-616
Troubleshooting PinToPin Utility 14-616
PinToPin Command Line 14-616
PinToPin Linux Compatibility 14-617
PinToPin Window Utility 14-618
Nets Tab 14-618
Extents Tab 14-620
Components Tab 14-622
Analysis Tab 14-624
Options Tab 14-628
PintoPin XML Interface 14-629
Basics 14-629
Stackup Modification 14-633
Component Models 14-635
Pin Groups 14-636
Surface Roughness 14-637
PintoPin Environment Variables 14-638

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

Network Data Explorer 14-638
Network Data Explorer Overview 14-639
NDE Ribbon 14-640
Network Data Selection Pane 14-641
Cell and Frequency Selection Pane 14-641
Data View Pane 14-642
Loading Data into Network Data Explorer 14-642
Exporting Data from Network Data Explorer 14-643
Exporting SYZ Data 14-643
Exporting Macromodel 14-646
Comparing Original S-Parameters with Exported S-Parameters 14-653
Creating an NPort Model 14-653
Scripting for Network Data Explorer 14-654
Network Data Explorer Ribbon 14-654
Network Data Explorer Data Sources 14-655
Network Data Explorer Display Format 14-655
Network Data Explorer Display Full Port Names 14-656
Network Data Explorer Save or Reset Default Settings 14-657
Smoothing 14-657
Cell Filtering 14-658
Changing Port Properties and Reducing Matrix Size 14-659
Displaying Mixed-Mode Parameters using Differential Pairs 14-660
Reset All Port Properties 14-662
Data View Pane Context Menus 14-662
Multiple Frequency Statistics 14-662
Highlight Min/Max 14-663
Select Transpose 14-663
Color Legend Attributes 14-664

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

Matrix Entries Plot Menu 14-665
Exploring Network Data and Modifying the Display 14-669
Viewing a Matrix Table 14-670
Viewing a Color-Coded Matrix Plot 14-671
Displaying a Cell Graph Across All Frequencies 14-672
Displaying Matrix Statistics by Frequency 14-673
Displaying Individual Statistics for All Frequencies 14-674
Creating a Statistics Plot 14-675
Comparing Network Data 14-676
Causality Checking and Plots 14-677
Multithreading 14-684
15 - Running Simulations 15-1
Solving a Single Setup 15-2
Running More than One Simulation 15-3
Monitoring Queued Simulations 15-3
Monitoring the Solution Process 15-4
Changing a Solution Priority for System Resources 15-5
Aborting an Analysis 15-6
Re-solving after Modifying a Design 15-6
Re-solving after Ansys Workbench Thermal Feedback 15-6
16 - Ansys Workbench Integration Overview 16-1
Integrating Ansys EM Suite with Ansys Workbench 16-1
Workbench Data Integration Overview 16-3
Adding New Analysis Systems 16-5
Importing Existing Ansys EM Projects into Workbench 16-7
Editing Ansys EM Models in Workbench 16-8
Analyzing Ansys EM Models in Workbench 16-9
Performing Parameter Studies in Workbench 16-11

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

Scripting in Workbench 16-15
Ansys EM - Ansys Multiphysics Coupling 16-16
Multiphysics Coupling on Workbench with Ansys Thermal 16-16
Multiphysics Coupling on Workbench with Ansys Structural 16-17
Multiphysics Coupling between Ansys EM Field Systems on Workbench 16-18
Ansys EM CAD Integration through Workbench 16-19
CAD Integration Functionality 16-22
CAD Integration and Geometry Sharing 16-24
Bi-Directional CAD Integration 16-27
CAD Integration Model Edits 16-28
Multiple Geometry Links for CAD Integration 16-30
Creating Dynamic Links to EDT Designs 16-30
Healing with CAD Integration 16-31
Important Geometry Options for CAD Integration 16-31
Ansys EM to Ansys Geometry Transfer 16-33
Material Assignment Transfer 16-34
Geometry Transfer through Ansys DesignModeler (DM) 16-35
Workbench Material Data Transfer 16-36
Detailed Behavior 16-36
Stress Feedback to HFSS using Workbench 16-37
Interface Changes for Stress Feedback from Ansys Mechanical to HFSS 16-38
Process Flow for HFSS Workbench for Stress Feedback 16-41
Only Stress Feedback 16-46
Feedback Iterator 16-46
The Feedback Iterator System 16-48
Feedback Iterator in Use 16-49
Feedback Iterator Component Properties 16-50
Feedback Iterator User Interface Operations 16-52

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

Callback Interface ... 16-53
Callback and State API ... 16-53
Utility Functions ... 16-55
Output/Debugging Functions ... 16-55
Example Scenarios for Feedback Iterator ... 16-55
Setting up Iteration with Feedback Iterator ... 16-56
Breaking Iteration Control ... 16-57
Starting an Iterative Update ... 16-58
Running a Single Iteration ... 16-59
Interrupting an Iterative Loop ... 16-59
Resuming an Interrupted Iterative Loop ... 16-59
Modifying the Systems Involved in Iteration (Coupled Clients) ... 16-59
Iterating to Convergence ... 16-59
Surface Force Density in HFSS ... 16-62
17 - Meshing In HFSS 3D Layout ... 17-1
Introduction to Meshing in HFSS 3D Layout ... 17-3
Meshing Method ... 17-4
Meshing With Rigid-Flex Bends ... 17-5
Initial Meshing Method ... 17-6
Access the Mesh Settings Window ... 17-7
Defining a Curved Surface Mesh Operation ... 17-9
Normal Deviation Angle ... 17-10
Maximum Surface Deviation Length ... 17-11
Aspect Ratio ... 17-12
Surface Approximation ... 17-13
Maximum Surface Deviation (Length) ... 17-17
Maximum Normal Deviation (Angle) ... 17-18
Aspect Ratio ... 17-19

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

Meshing for Curved Surfaces 17-21
Model Resolution Length 17-21
Auto Option 17-22
An MRL Scenario 17-23
Operations Refining Initial Mesh 17-26
Mesh Setup for Mesh Fusion/Assembly 17-26
Activating the Tau Mesher Beta Feature 17-29
Pre-Processing Settings for Components 17-33
Creating Mesh Regions and Activating Parallel Refinement (IC Mode Only) 17-33
Editing the Mesh Region 17-45
Mesh Generation 17-48
Generating Mesh without Solving 17-49
Adaptive Mesh Refinement 17-50
Adaptive Mesh Refinement Process Flow 17-53
Stages of Meshing 17-54
Choose a Solver 17-58
Assign Mesh Operations 17-59
Revert to Initial Mesh 17-59
Create Mesh Plot 17-60
HFSS 3D Layout Mesh Seeding 17-61
Adding Length-Based Mesh Seeding to an HFSS 3D Layout Design 17-61
Adding Skin-Depth Mesh Seeding to an HFSS 3D Layout Design 17-63
Seed Refinement for Mesh Growth 17-66
Mesh Refinement Scenarios 17-67
Assign Mesh Operations Upfront 17-67
Restrict Length of Elements 17-68
Seed Mesh by Region 17-71
The Phi Mesher 17-73

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

Meshing With the Maximum Delta-Z Ratio 17-74
The Mesh Feedback Window 17-75
Phi Mesher for Multilayer Complex Package Layout 17-77
The Phi Plus Mesher 17-79
Selecting the Phi Plus Mesher 17-80
Enhanced Phi Plus for ECAD/MCAD Assembly 17-83
HFSS Mesh Simplification 17-84
HFSS Mesh Errors 17-90
Troubleshooting Mesh Failures 17-91
18 - HFSS 3D Layout Technical Notes 18-1
The MPIE Method 18-1
Representation of the Current 18-2
Method of Moments 18-2
Size of Mesh Versus Accuracy 18-3
Loop Tree Discretization 18-3
MPIE Method Implementation 18-4
The Adaptive Analysis Process 18-4
Ansys Fast Solver Technology 18-6
Broadband Fast Frequency Sweeps 18-6
EMI Scanner Technical Notes 18-7
EMI Scanner Rule Setting Notes 18-7
Component Pin Grid Settings 18-7
Decoupling All Plane Combinations 18-8
I/O Net Classes 18-8
HFSS 3D Layout S-Parameters 18-9
S-Parameters and Frequency Points 18-9
Computing Characteristic Impedance 18-10
Renormalized S-Matrices 18-10

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

Computing the Z-Matrix ... 18-12
Computing the Y-Matrix ... 18-12
Deembedded S-Matrices ... 18-12
HFSS 3D Layout Modes ... 18-13
Modes, Reflections, and Propagation ... 18-14
Modes and Frequency ... 18-14
Evanescent Modes and Attenuation ... 18-14
Viewing the HFSS 3D Solution Log File ... 18-14
Exporting S-Matrices for Full-Wave SPICE Analysis ... 18-15
Frequency-Dependent Material Properties ... 18-15
Obtaining Broadband Matching between Huray and Hammerstad Models ... 18-16
Obtaining Better Low Frequency Matching ... 18-18
Modified Hammerstad Model ... 18-20
Djordjevic-Sarkar Causal Dielectric Model ... 18-21
19 - Scripting ... 19-1
20 - Glossary ... 20-1
Glossary: A ... 20-1
Glossary: B ... 20-4
Glossary: C ... 20-6
Glossary: D ... 20-10
Glossary: E ... 20-14
Glossary: F ... 20-17
Glossary: G ... 20-20
Glossary: H ... 20-21
Glossary: I ... 20-23
Glossary: J ... 20-25
Glossary: K ... 20-26
Glossary: L ... 20-27

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

Glossary: M 20-29
Glossary: N 20-32
Glossary: O 20-35
Glossary: P 20-37
Glossary: Q 20-41
Glossary: R 20-41
Glossary: S 20-43
Glossary: T 20-49
Glossary: U 20-52
Glossary: V 20-53
Glossary: W 20-54
Glossary: X 20-55
Glossary: Y 20-56
Glossary: Z 20-56
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 HFSS 3D Layout Help

HFSS 3D Layout helps you design multi-layer structures such as millimeter-wave integrated circuits (IC), micro-wave integrated circuits, printed circuit boards, planar antennas, and more by simulating their layout geometry. HFSS 3D Layout simulates layer structures of high-frequency electromagnetic fields through an innovated meshing technology called Phi mesher. The Phi mesher is an advanced, layout-based meshing tool that is capable of rapidly generating an initial mesh. This initial mesh facilitates faster simulations that can be further enhanced and accelerated using many of the various available adjuncts of high performance computing. You can take advantage of a closer integration between the 3D Layout and external data sources by using Layout Links, Cadence APD, and SPB.

HFSS 3D Layout is suitable for aiding in the design of the following multi-layer structures:

- Millimeter-wave integrated circuits
- Micro-wave integrated circuits
- Printed circuit boards
- Planar antennas

Before using HFSS 3D Layout, you can become familiar with it through getting started guides that are provided with the Ansys Electronics Desktop and its software systems such as Layout Editor, HFSS, and other tools for circuit and system simulation. This chapter describes the basics of using HFSS 3D Layout to design and simulate multi-layer structures for various materials, models, and excitations.

HFSS 3D Layout has three getting started guides to help you learn this product:

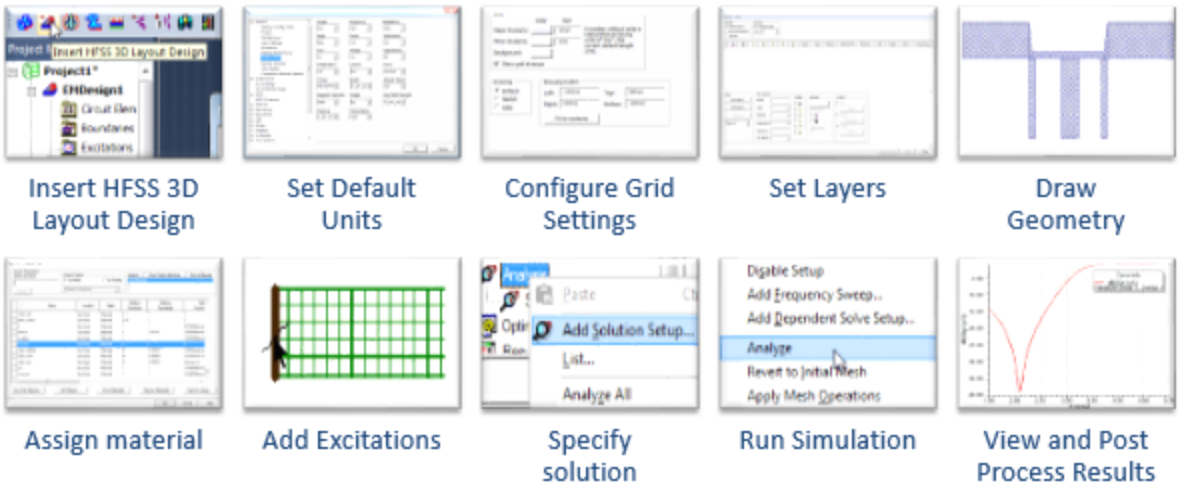
- *Getting Started with HFSS 3D Layout: Low Pass Filter*
- *Getting Started with HFSS 3D Layout: Microstrip Filter*
- *Getting Started with HFSS 3D Layout: Slot Fed Patch Antenna*

Workflow of an HFSS 3D Layout Design

Setting up an HFSS 3D Layout design is very similar to setting up an HFSS design or an HFSS-IE design. To set up an HFSS 3D Layout 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.

Defining a solution setup enables HFSS 3D Layout to compute a solution for a design. You can define multiple solution setups for the same design. If you right-click in on the **Project Manager** window and select **Add Nexxim Solution Setup** , select a type of analysis to set up, which opens a solution setup window. You can specify multiple settings in the solution setup window for computing a solution for a 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. Insert an HFSS 3D Layout design into a project.
2. Draw the model geometry, specifically:
    a. Create the metal and dielectric layers of the design.
    b. Assign a thickness, roughness, elevation, and material to each layer.
    c. Draw the geometry of the signal layers.
3. Assign material characteristics
4. Define the ports or incident waves in the model.
5. If applicable, insert an external device, an *N-port*, with known *S*-parameters into your design.
6. Specify how to compute the solution by adding a solution setup.
7. (Optional) Set up any Optimetrics you want to run.
8. Run the simulation. HFSS 3D Layout computes the current inside the structure.
9. View the solution results, post-process results, and reports, and create field overlays.

Note:

Some of these tasks may not be applicable to your designs. Not all designs require *N*-ports.

Note:

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

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

Recording a Script to a File

Once users start to record a script, their subsequent actions are added to the script. Each interface command has one or more associated script commands that are recorded to the script. The script is recorded to a text file in .py (Python) format.

Follow these steps to record a script to a file.

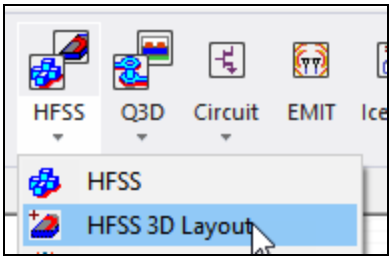
1. Click **Tools** > **Record Script** to open an explorer window.
2. Navigate to a selected directory to save the script. Then double-click the folder's name.
3. Type the name of the script in the **File name** text box.
4. Click **Save**.
5. The script is saved in the folder you selected by the file name (e.g., *filename*.py).
6. Perform the steps that you want to record.
7. When you have finished recording the script, click **Stop Script Recording** on the **Tools** menu.

Inserting an HFSS 3D Layout Design

This page briefly describes how to set up and run an HFSS 3D Layout design in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. The focus is on circuit and system analysis and the framework required to design RF-microwave, signal integrity, and printed circuit board applications. Before beginning, launch **Electronics Desktop** (**Electronics Desktop**), create a new project, and also enable the display of the **Choose Technology** window.

To insert an HFSS 3D Layout design:

1. From the **Project** menu, select **Insert HFSS 3D Layout Design** or in the **Desktop** ribbon, open the menu under the HFSS icon and select **HFSS 3D Layout** to open the **Choose Layout Technology** window.



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

So long as the check box **Show this window on new design** is selected, this window appears when a new design is inserted into a project. If the check box is cleared, the next time that a new design is inserted, the window does not appear and a design is inserted using the selected technology on the last insertion.

1. Select an existing EM technology from the list or click **None** to manually add the model's metal and dielectric layers.

   A new design appears in the **Project Manager** window. The design is named HFSS3DLayout*n* by default, where *n* is the order of the design in a potential series of designs. Folders named **Data**, **Excitations**, **Analysis**, **Optimetrics**, and **Results** automatically appear under the design name. These folders store data about the design.

   To automatically insert an HFSS 3D Layout design when creating a new project, see **General Options** > **Desktop Configuration** to open the **Layout** view window. You can now create or edit the design layers in the **Edit Layers** window.

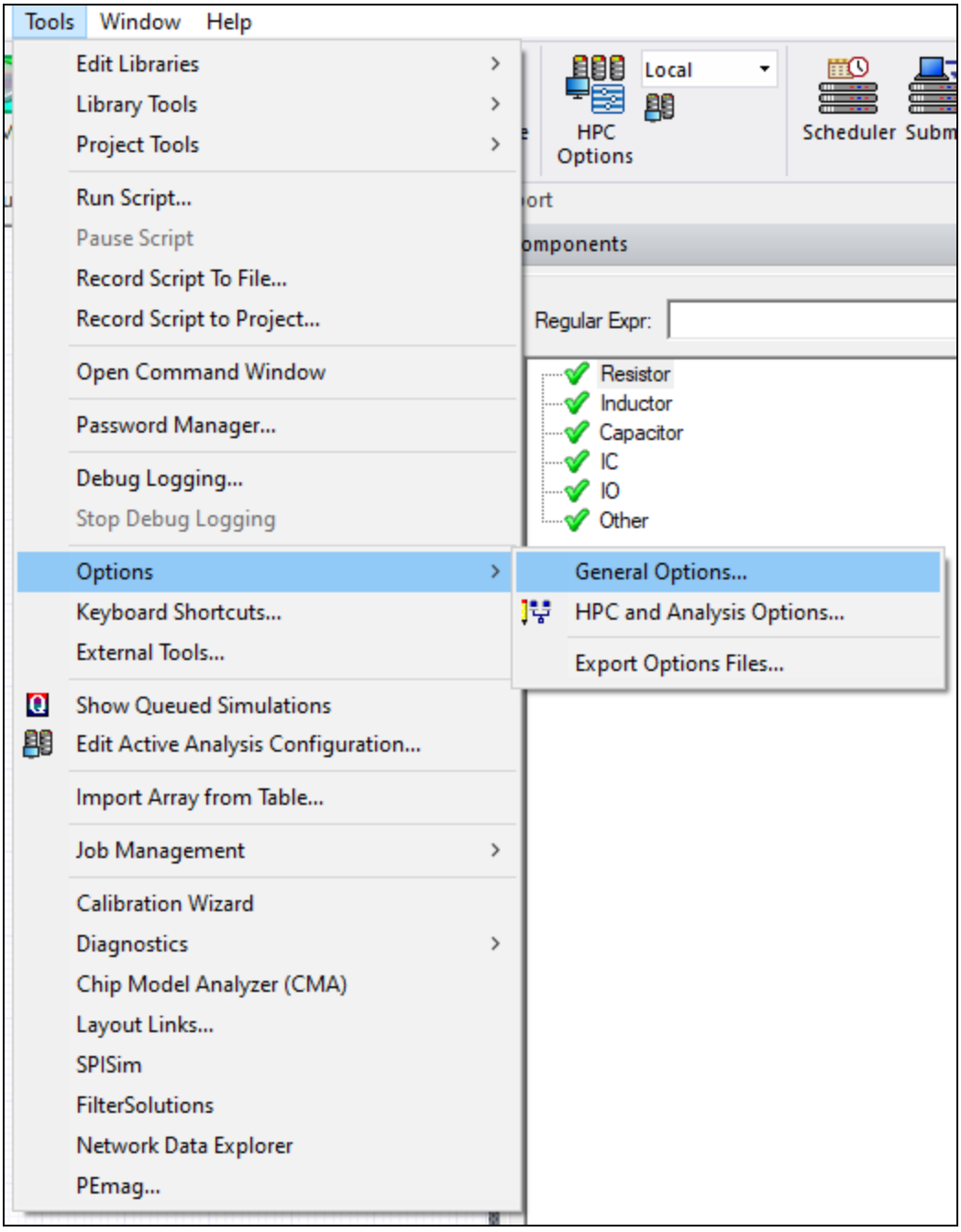
> **Note:**
>
> To expand the **Project Tree** and view specific data about the model such as its boundary assignments, click **+** to the left of the design icon in the **Project Manager** window.

Inserting an EM Technology File

HFSS provides default stackups for common structures — *technology files*. You can create and save your own stackup as a technology file so that it is available to every project. Each technology file you create includes the layers' attributes: material property, units, thickness, elevation, and roughness data. This information is stored in an .asty file in the **\userlib** folder. Follow these steps to enable the **Choose Technology** window and then select an EM technology.

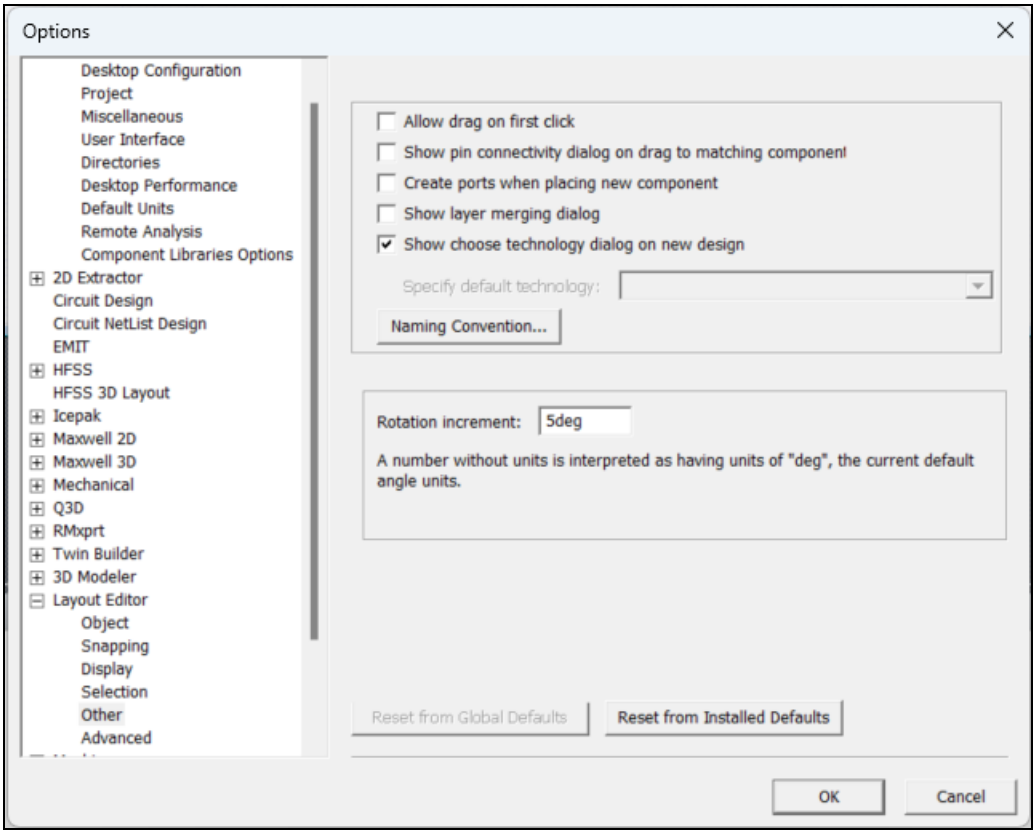
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window.

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



2. Expand **Layout Editor** and select **Other**. Then check the **Show choose technology dialog on new design** box.

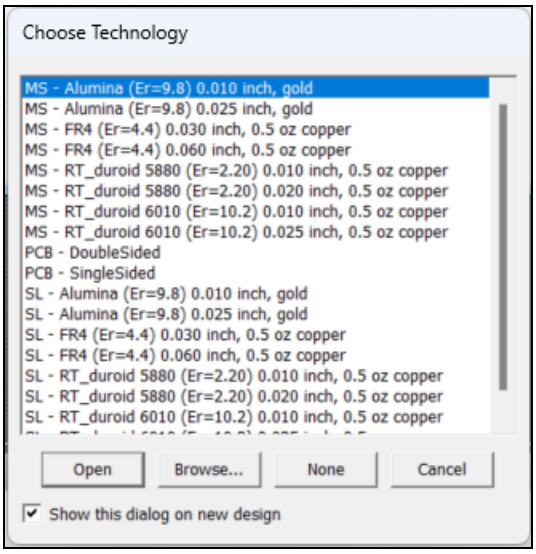
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **OK** to close the **Options** window.
4. Insert a new design **HFSS 3D Layout** design in a project to open the **Choose**

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

Technology window.



5. Select a structure from the list.

> **Note:**
>
> Alternatively, click **Browse** to search for an existing **.asty** technology file.

6. Click Open. The technology file you selected is listed in the **Project Tree** as EM*n* by default, where *n* is the order of the design in a potential series of designs. Sublevels named **Data**, **Excitations**, **Analysis**, and **Results** are automatically created underneath the design name in the **Project Tree**, storing data about the current project. The **Layout** view window also appears. You may now create or edit the design layers.

> **Note:**
>
> To import a technology definition from a technology definition file, refer to Adding a Technology Definition File to a Design (IC Mode only).

HFSS 3D Layout Options

Configure global changes to HFSS 3D Layout on the **Tools** > **Options** > **General Options** window. Make choices in the **General Options** window to create time-saving configurations for all projects at once. This greatly aids in the setup and execution of solution files and the data files which are produced. The specified options are used for all projects, unless explicitly overridden in a design.

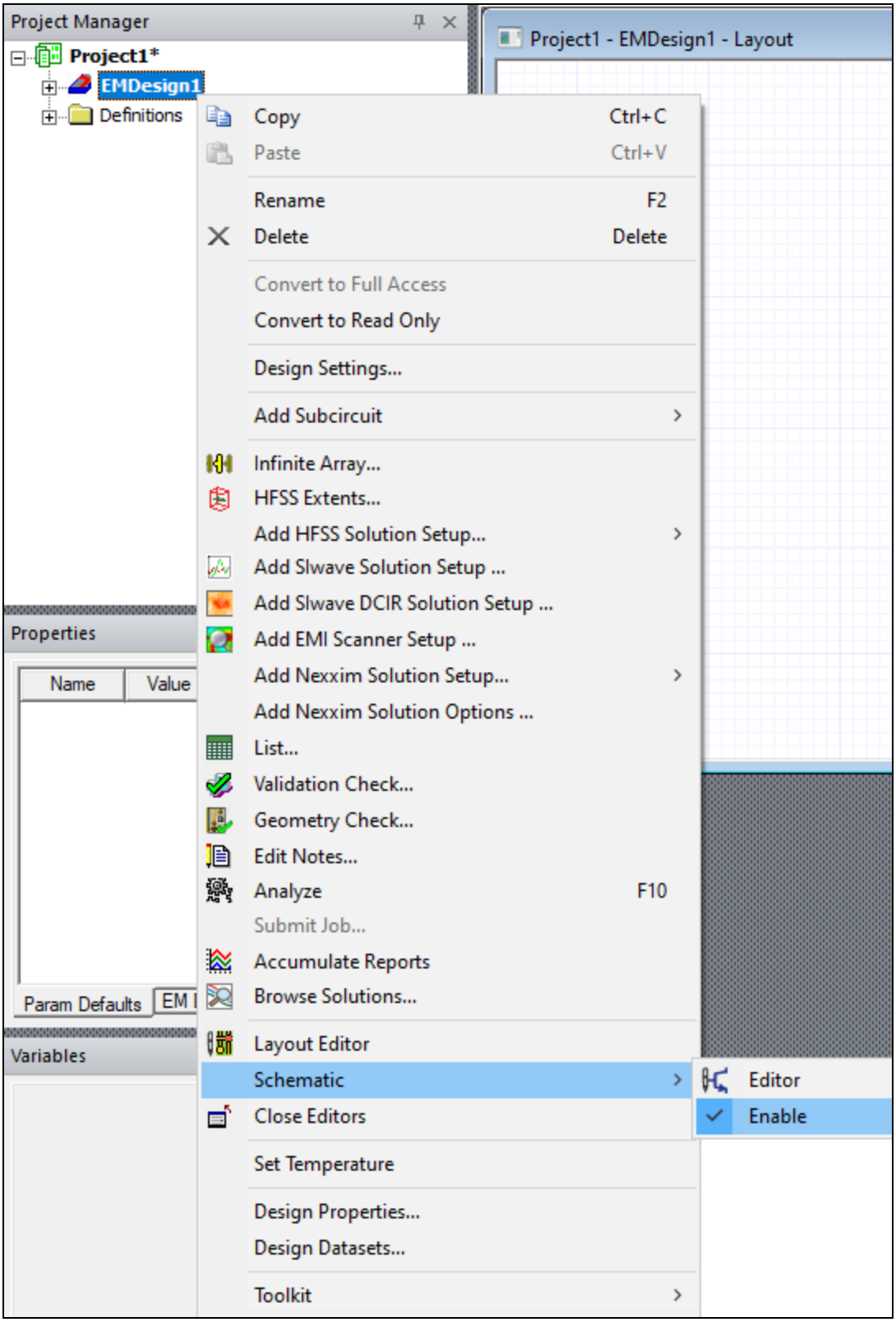
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **General Options** window, select **HFSS 3D Layout** to configure the following options in the **Interface Options** area.

- Choose whether to **Save solution files in binary mode for new projects.** Enabled (checked) by default.
- **Enable Schematic** controls whether new HFSS 3D Layouts, and designs imported from ANF files, have schematics activated or deactivated. When **Enable Schematic** is unchecked (deactivated), all **Schematic Editor** menu/toolbar options are unavailable. Enabled (checked) by default.

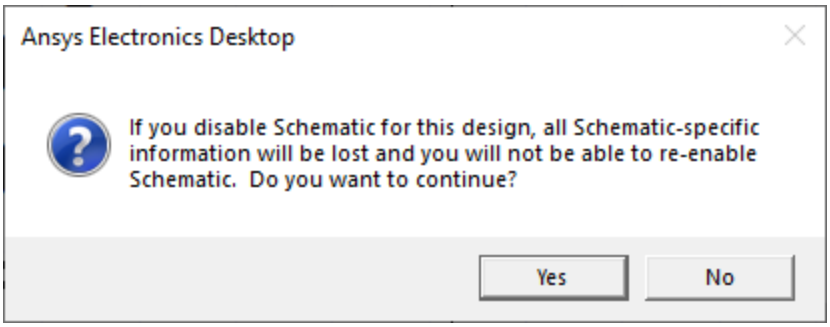
  To deactivate schematic options for an active design, first expand the **Project Tree** in the **Project Manager** window. Then right-click the active design and select **Schematic** > **Enable** to remove the check mark and deactivate schematic options.

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.

If **Enable Schematic** is unchecked, the following warning message appears. Once it is deactived, **Enable Schematic** cannot be re-enabled for a design.



It is possible to have a project in which some designs have schematics enabled and other designs have schematics disabled. Objects **cannot** be copied from designs without schematics into designs with schematics. Objects **can** be copied in the opposite direction but the copy operation does not contain schematic-specific information, and the paste option is deactivated.

- Choose whether to **Save before solving**. Enabled (checked) by default.

  If **Save before solving** is checked (enabled), and a major edit is made, the project is saved if a solve is started. Also, if a new project is created and a solve is started, the **Save As** window opens automatically and prompts users to save the project before continuing. If this option is unchecked, the user is not prompted to save before a solve.

  > **Note:**
  >
  > If users do not have permissions to write to the project folder, or if one of the files in the project folder is read-only, then choosing **Save** or **Analyze** prompts users to save the project to another location. Canceling the **Save As** cancels the Analysis.

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

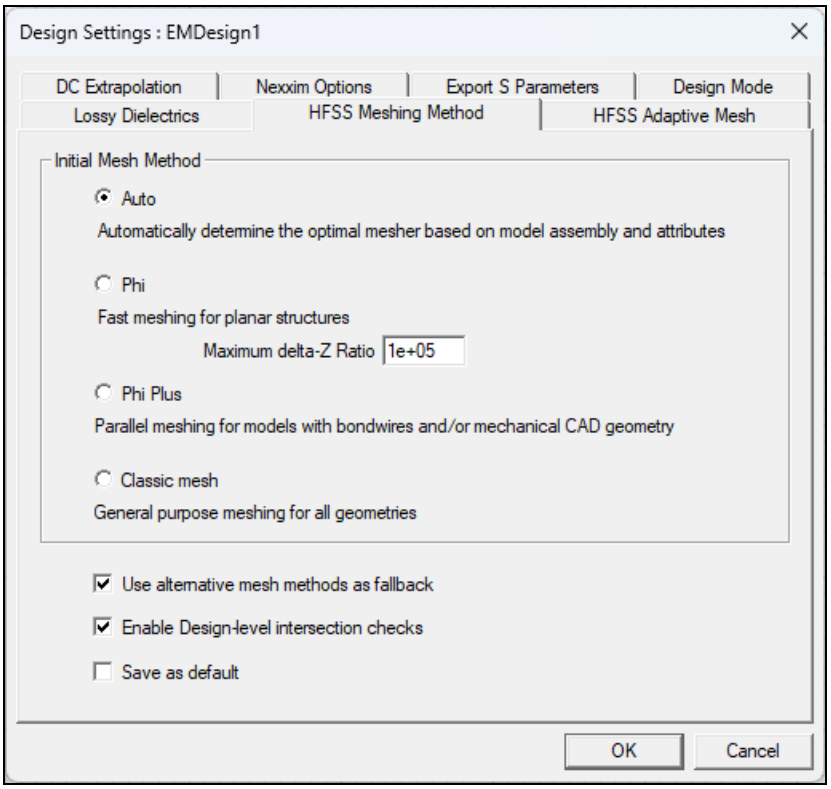
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- IcepakFEA
- Q3D
- 2D Extractor

HFSS 3D Layout Design Settings

From the **HFSS 3D** menu, select **Design Settings** to display the **Design Settings** window for the active design. The Design Settings window has seven tabs:

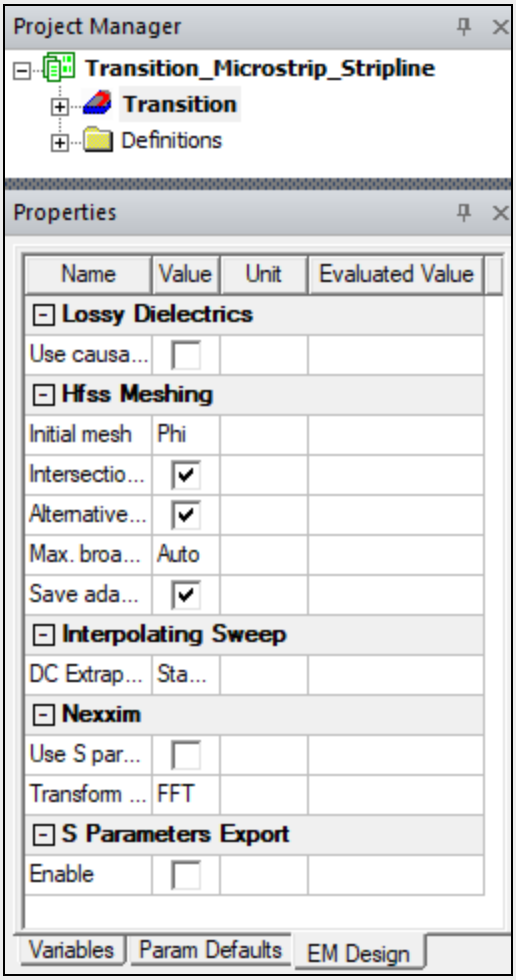
- DC Extrapolation
- Nexxim Options
- Export S-Parameters
- Design Mode
- Lossy Dielectrics
- HFSS Meshing Method
- HFSS Adaptive Mesh



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

Note:

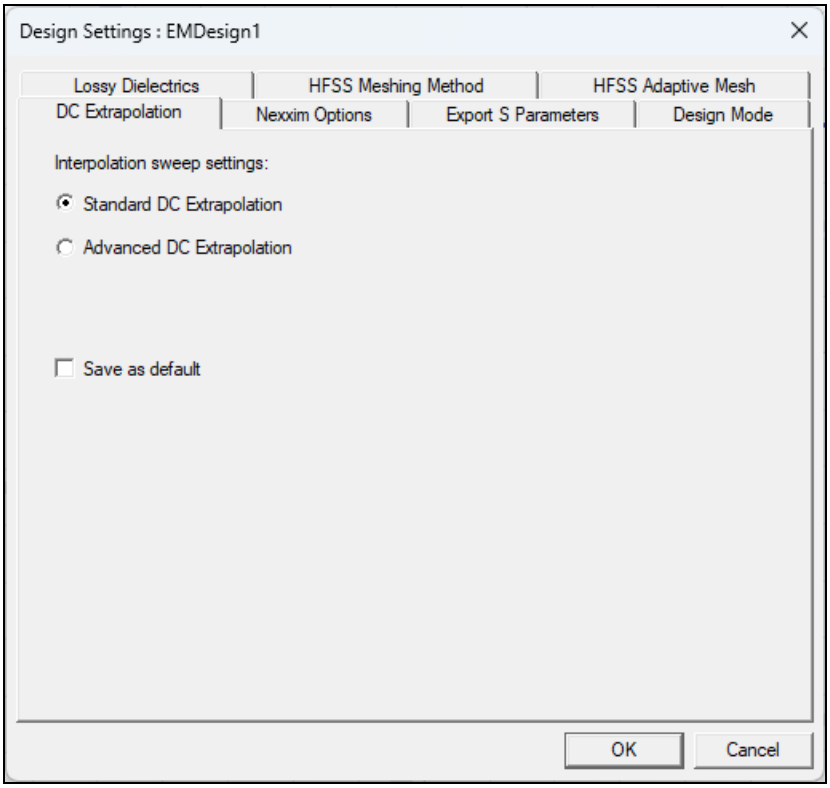
Selecting **[*Active Design Folder*]** in the **Project Tree** also displays **Design Settings** in **Properties** window under **EM Design** tab.



DC Extrapolation Tab

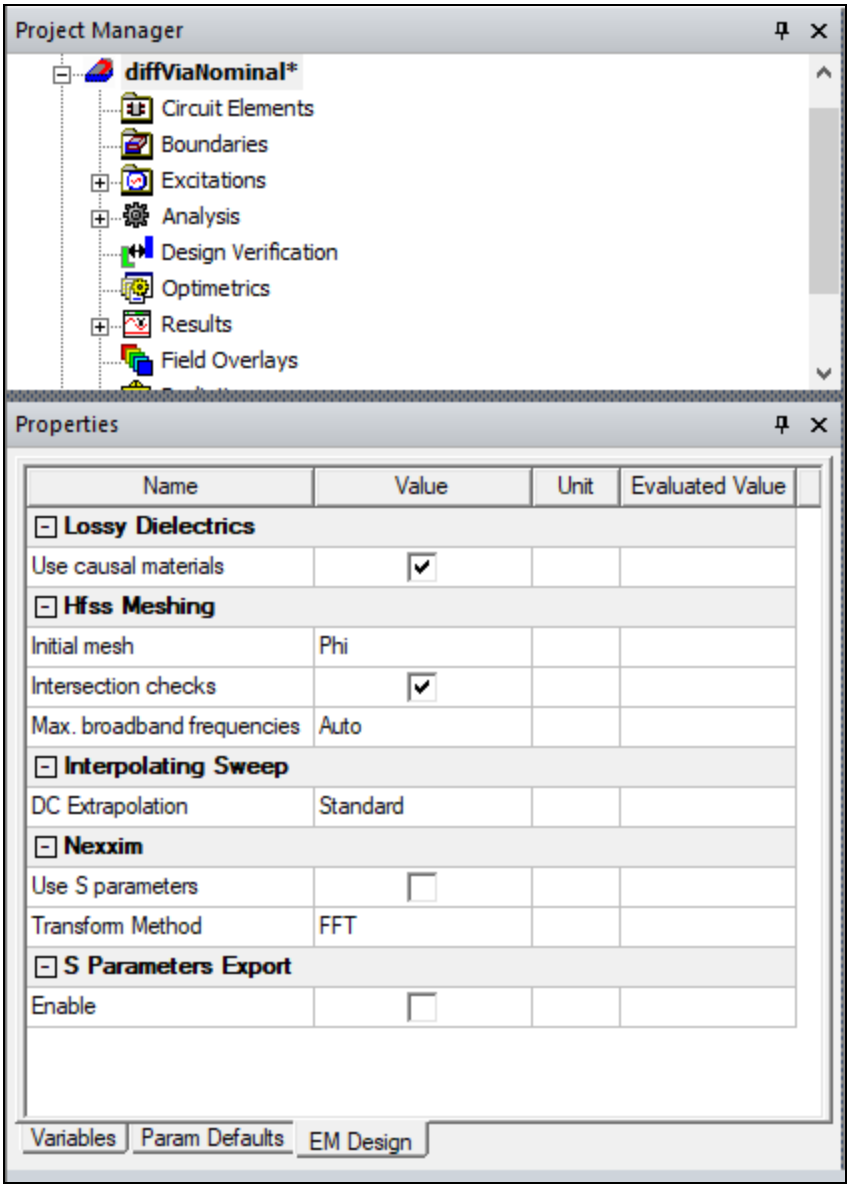
From the **Project Manager** window, right-click on a design and select **Design Settings** to open the **Design Settings** window and click the **DC Extrapolation** tab.

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



Also view and edit design settings by selecting a design using the **EM Design** 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.



- **Standard DC Extrapolation** causes the software to compute values automatically and hides the **DC Extrapolation** tab in the **Interpolating Sweep** setup.
- **Advanced DC Extrapolation** allows users to set a **Minimum Solved Frequency**.

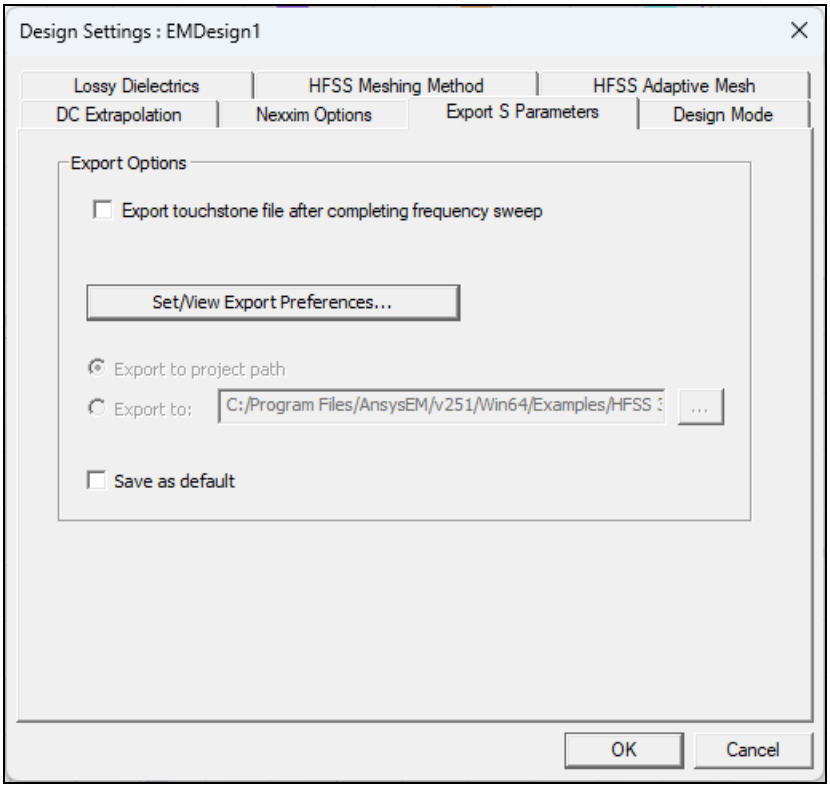
> **Important:**
>
> Selecting **Advanced DC Extrapolation** is **not** recommended by Ansys and exists to support legacy functions. Selecting **Advanced DC Extrapolation** may have unexpected results.

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

- Select **Save as default** and click **OK** to save these setting to the registry.

Export S-Parameters Tab

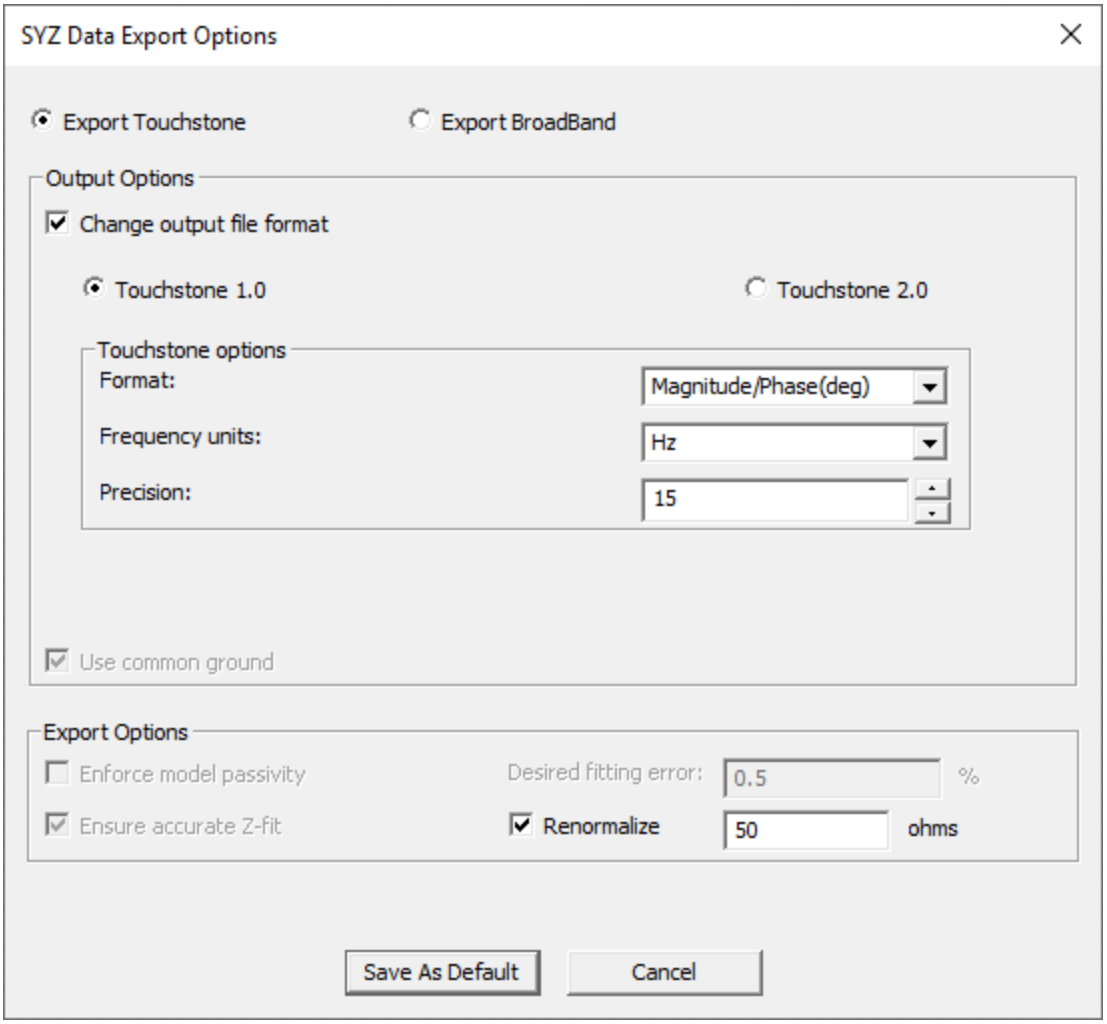
From the **Project Manager** window, right-click on a design and select **Design Settings** to open the **Design Settings** window and click the **Export S-Parameters** tab.



The **Export S-Parameters** tab lets you **Set/View Export Preferences** and specify whether to Export Touchstone file after completing frequency sweep.

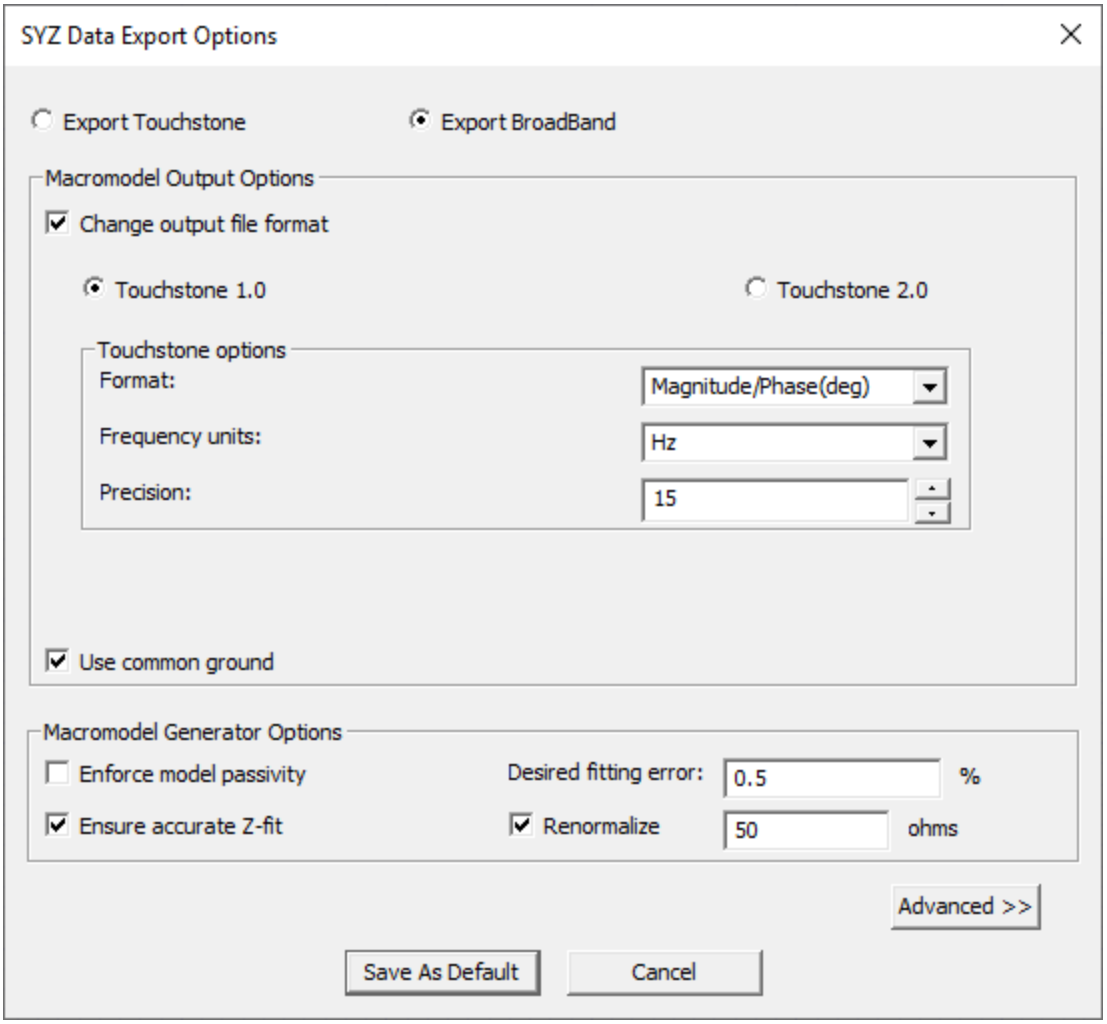
Click **Set/View Export Preferences** to open the **SYZ Data Export Options** window. This window contains a range of choices for output format, including advanced options. SYZ Data Export supports 2 types of export: touchstone export and broadband export.

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 touchstone export option, at the end of a simulation raw s-parameter data is exported either in Touchstone 1.0 or Touchstone 2.0 as selected by the user.

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

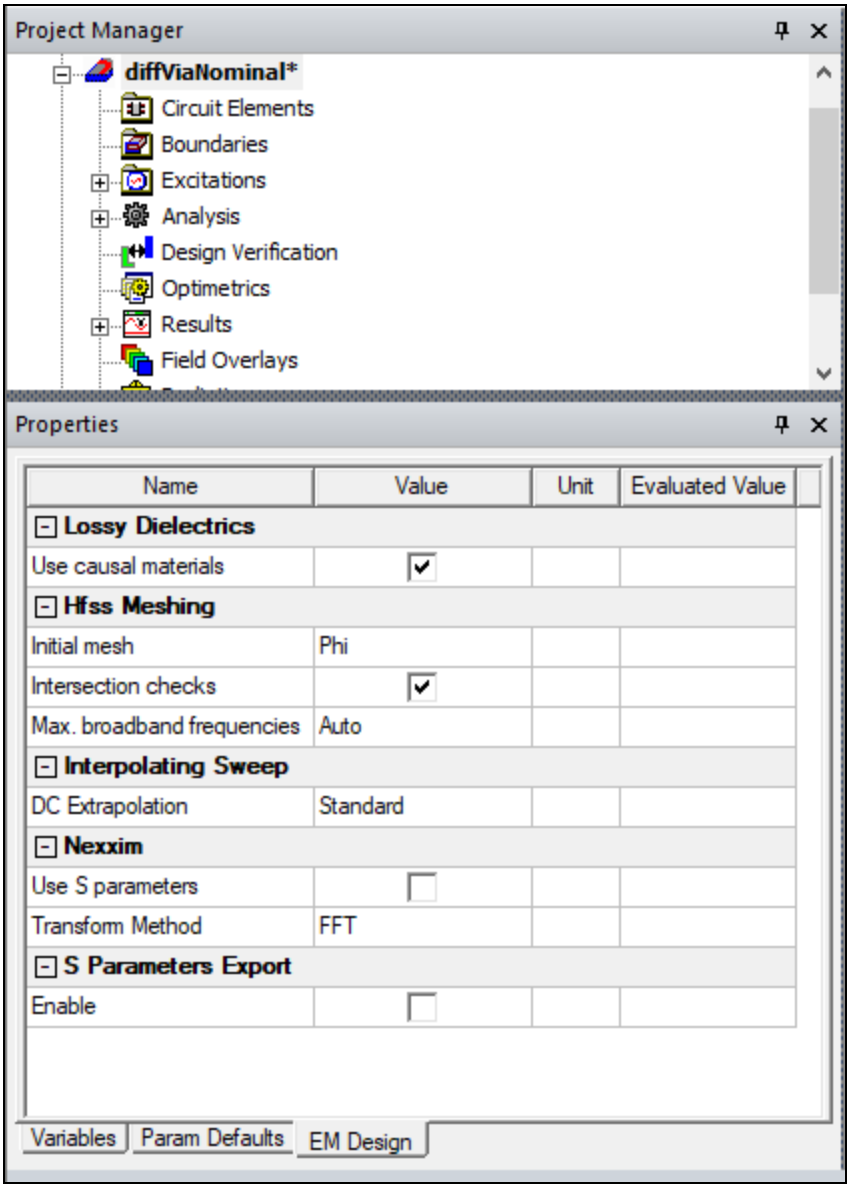


If you select the broadband option, at the end of a simulation the s-parameter goes through a fitting based on the export options selected and data is exported either in Touchstone 1.0 or Touchstone 2.0 as selected by the user.

Click **Save as default** to save the export settings to the registry.

Selecting a active design in the **Project Tree** also displays the **Design Settings** in **Properties** window under the **EM Design** tab.

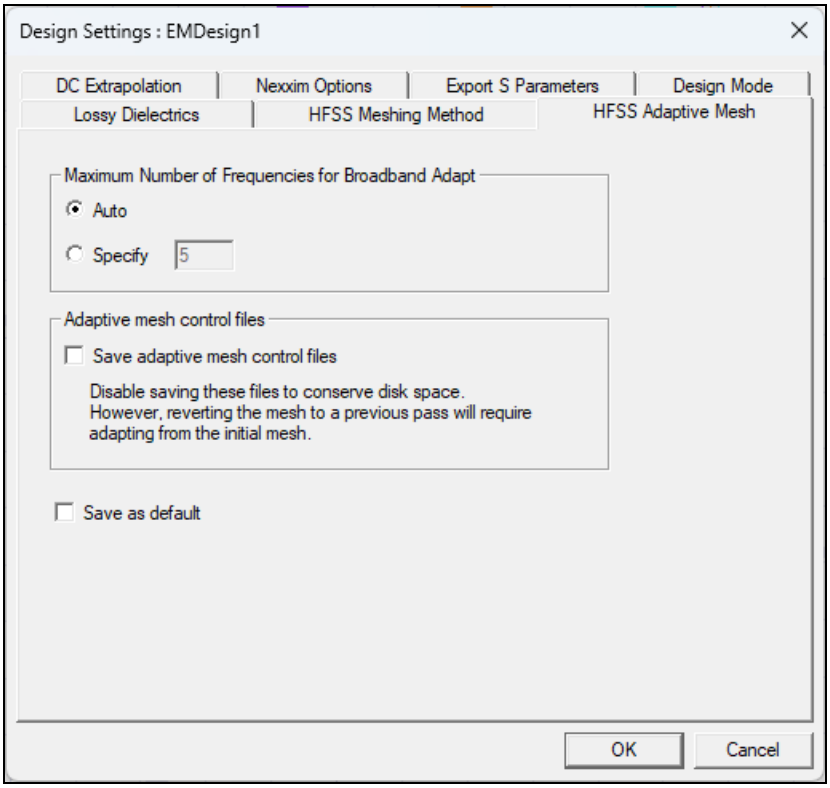
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



HFSS Adaptive Mesh Tab

From the **Project Manager** window, right-click on a design and select **Design Settings** to open the **Design Settings** window and click the **HFSS Adaptive Mesh** tab.

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

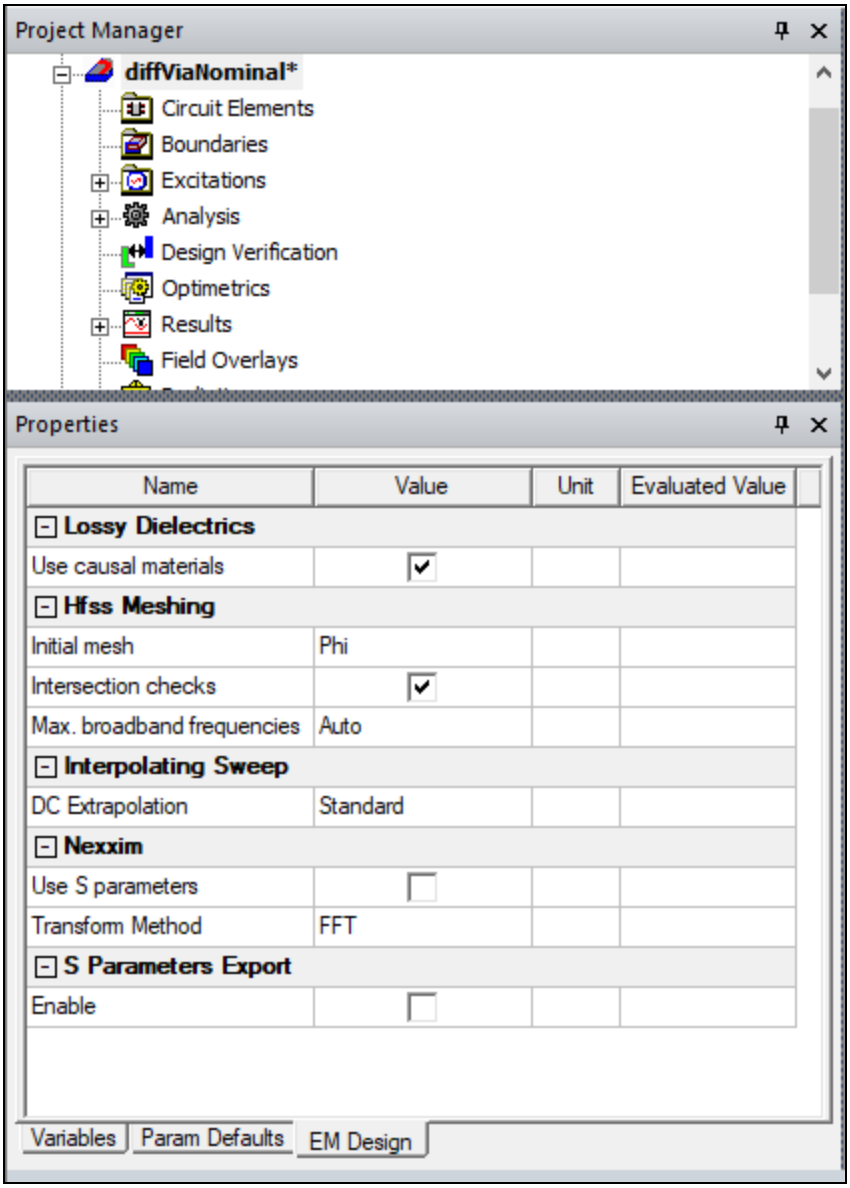


- **Maximum Number of Frequencies for Broadband Adapt**: Normally the decision for how many frequencies to use for is set automatically. Because the decision is not straightforward, this option is for advanced users to manually set the maximum number of frequencies to solve during broadband adaptive meshing. This feature works for both auto and advanced HFSS solution setups.
- **Save adaptive mesh control files**: Allows you to deactivate saving adaptive mesh control files to save disk space. However, if unchecked, reverting the mesh to a previous pass requires adapting on the initial mesh. Changing the setting reverts to the initial mesh.

- Select **Save as default** and click **OK** to save the setting to the registry.

Also view and edit design settings by selecting a design using the **EM Design** tab of the **Properties** window.

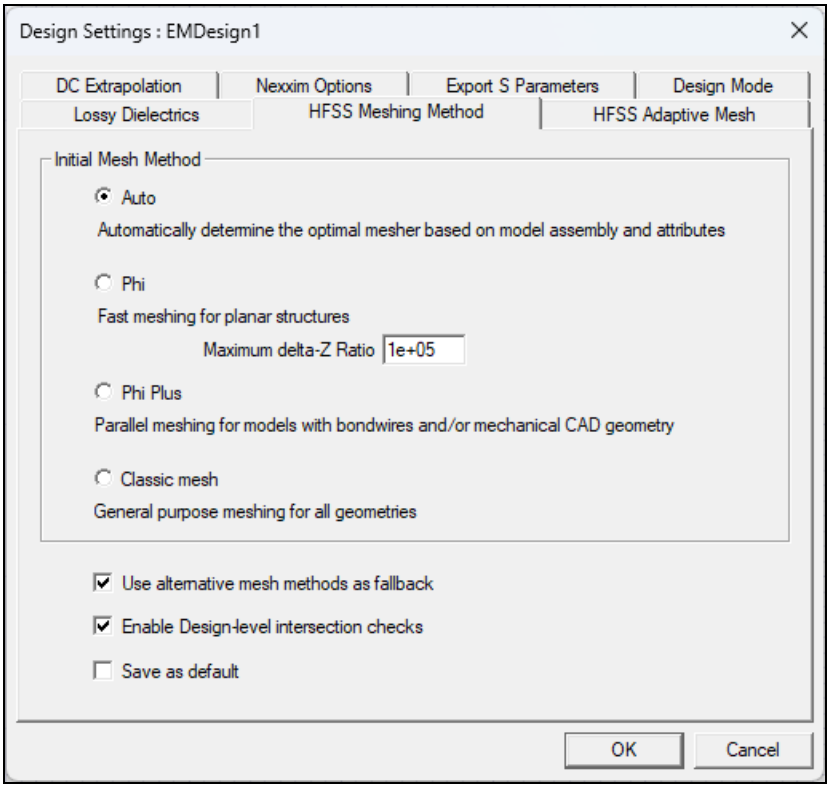
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



HFSS Meshing Method Tab

From the **Project Manager** window, right-click on a design and select **Design Settings** to open the **Design Settings** window and click the **HFSS Meshing Method** tab. Change the following settings, as necessary.

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

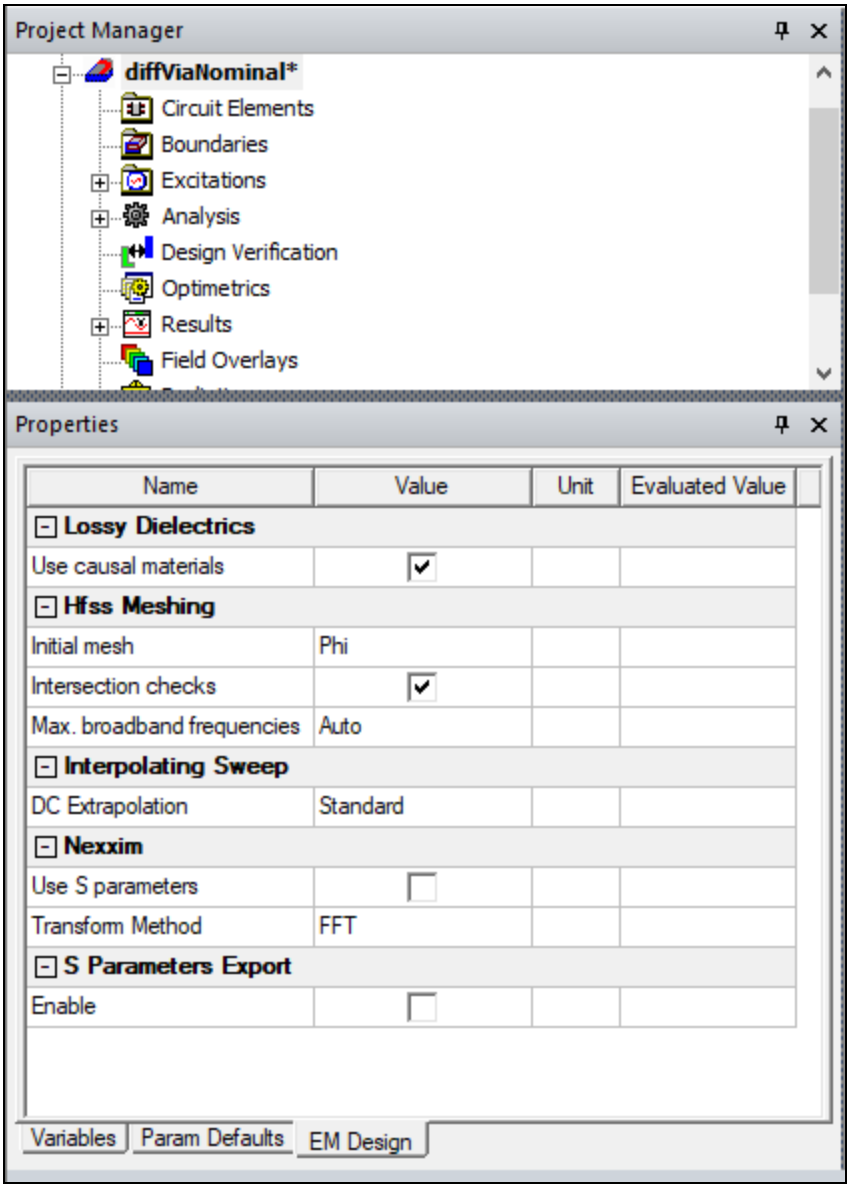


- From the **Initial Mesh Method** area, select one of the following mesh methods:
  - **Auto** — automatically selects a suitable mesh method for the design based on geometry characteristics (e.g., if the design contains bondwires, **Phi Plus** is selected).
  - **Phi** — a layout-based meshing technology, available in the HFSS 3D Layout interface. This advanced meshing technology is capable of rapidly generating an initial mesh ensuring faster simulations that can be further accelerated and enhanced by using high performance computing.
  - **Phi Plus** — works with all types of layout models, particularly those containing bondwires, solder balls, or 3D Components (MCAD). Phi Plus generates an initial mesh using high performance computing (HPC).
  - **Classic mesh** — uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately.
- Check the **Use alternative methods as fallback** box to allow the solver to use an alternative mesh method if the selected mesh method (e.g., Phi) fails to generate an initial mesh. Enabled 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.

- Check the **Enable Design-level intersection checks** box and the Ansys Electronics Desktop (AEDT)™ electronics systems design platform automatically searches for and notifies the user when a design has intersecting geometries.
- Check the **Save as default** box and click **OK** to save the causal material setting to the registry.

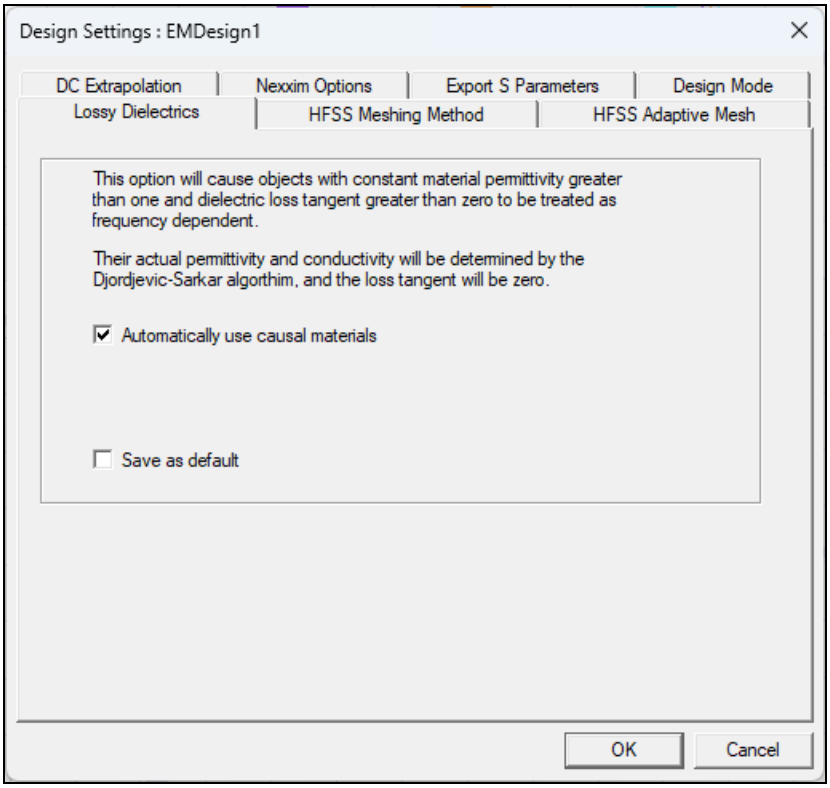
Also view and edit design settings by selecting a design using the **EM Design** tab of the **Properties** window.



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

Lossy Dielectrics Tab

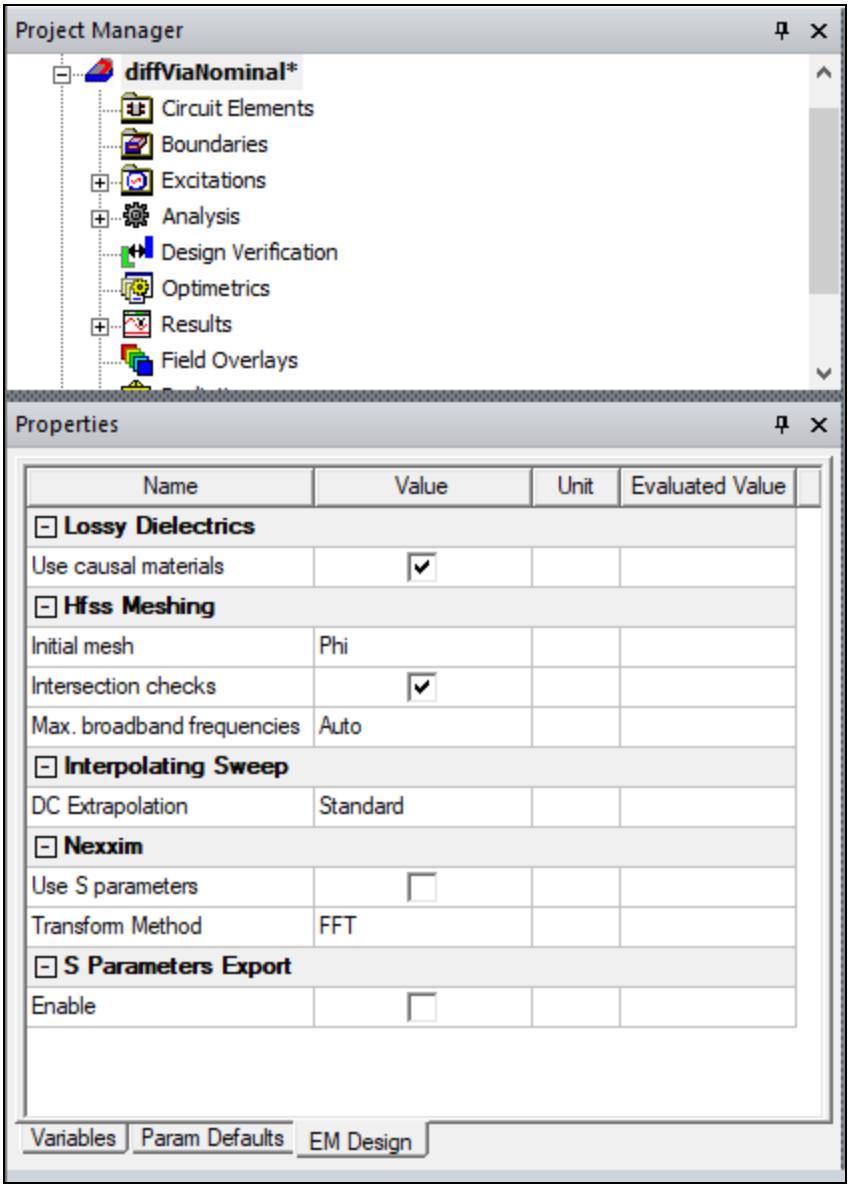
From the **Project Manager** window, right-click on a design and select **Design Settings** to open the **Design Settings** window and click the **Lossy Dielectrics** tab.



- **Automatically use causal materials** causes some objects to be treated as frequency dependent.
- Select **Save as default** and click **OK** to save the causal material setting to the registry.

Also view and edit design settings by selecting a design using the **EM Design** tab of the **Properties** window.

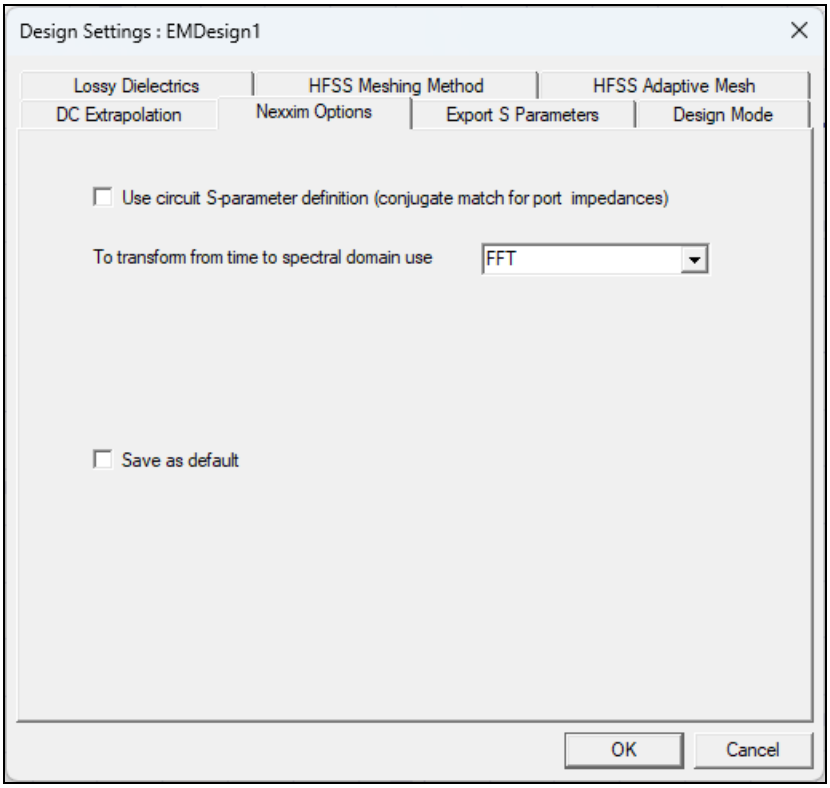
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Nexxim Options Tab

From the **Project Manager** window, right-click on a design and select **Design Settings** to open the **Design Settings** window and click the **Nexxim Options** tab.

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



The **Nexxim Options** tab allows you to configure options for Nexxim setups.

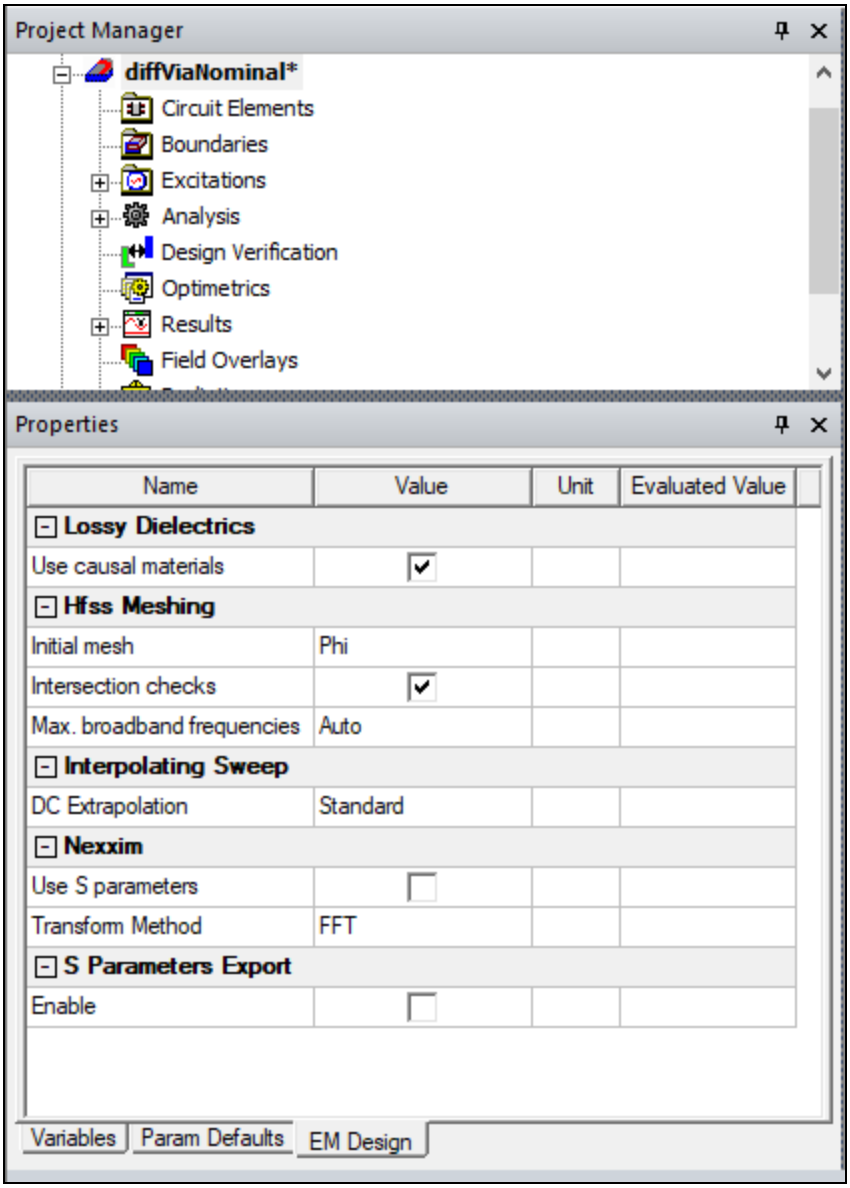
- **Use circuit S-parameter definition** — Controls the definition for port impedances. For HFSS 3D Layout, this option should typically be unselected (check box unchecked). The definition of a “matched port” depends on the application. In applications that deal mostly with circuit quantities (including the Nexxim simulator), a “matched port” has a characteristic impedance that maximizes the power transfer. This is also called a “conjugate match.” In applications that deal mostly with electromagnetic quantities (including HFSS), a “matched port” has a characteristic impedance that maximizes the transfer of the voltage wave. The two definitions differ only by the conjugate of the characteristic impedance of the port. For real port impedances, there is no difference.
- **To transform from time to spectral domain use** – Allows you to select the method used to transform on the time domain to the spectral domain for all projects.
    - **Fourier Integration** is very accurate but may be time-consuming for large datasets. The Fourier Integration method computes the projection of the signal on each of the requested harmonics via integration of the inner-product integral.
    - **FFT** (Fast Fourier Transform, the default) achieves faster conversion with a slight reduction in accuracy. The FFT method samples the signal at 2*(maximum harmonic number) points, and outputs the FFT of the sampled signal. The FFT

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

method may not be robust enough to avoid aliasing while converting some time-domain signals to the frequency domain.

- Select **Save as default** and click **OK** to save the causal material setting to the registry.

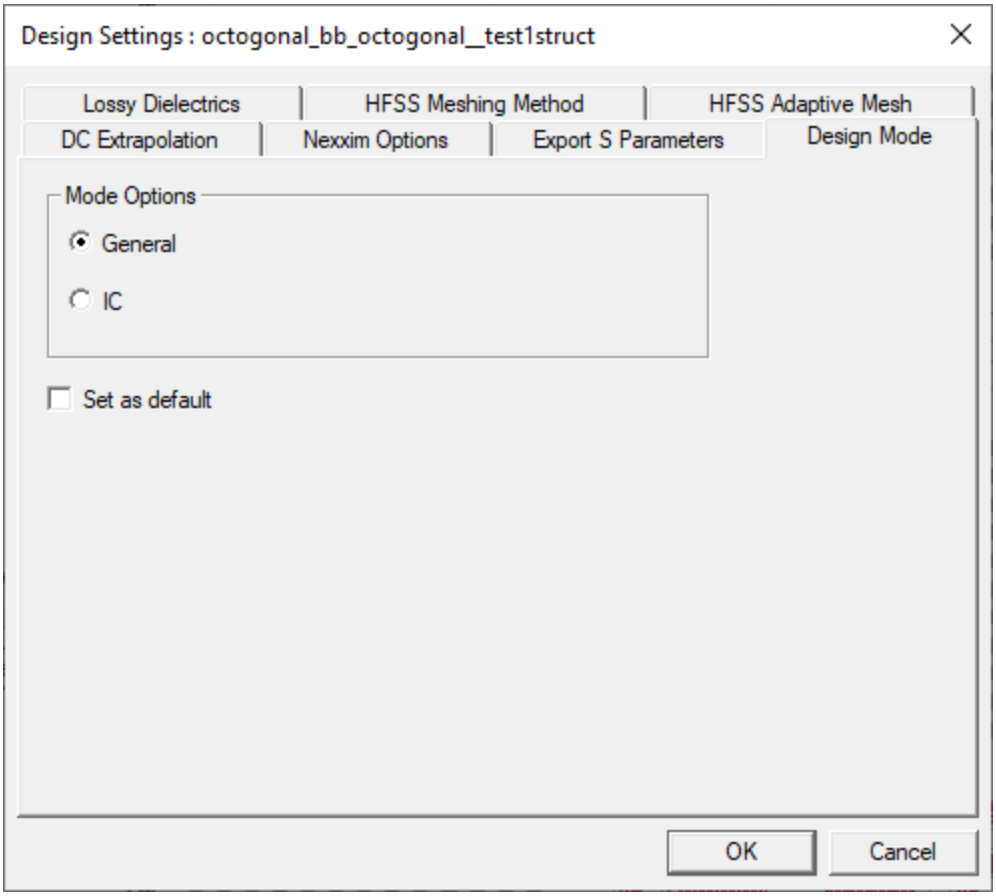
Selecting a active design in the **Project Tree** also displays the design settings in **Properties** window under **EM Design** tab.



Design Mode Tab

From the **Project Manager** window, right-click on a design and select **Design Settings** to open the **Design Settings** window and click the **Design Mode** 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:

Refer to Layout Editor and Switching to IC Mode for more information.

Editing Distributed Machine Configurations in HFSS 3D Layout

You can create a new distributed machine configuration, or edit an existing machine configuration, for Ansys SIwave™ and/or Ansys HFSS™.

Distributed Machine Configurations for HFSS 3D Layout

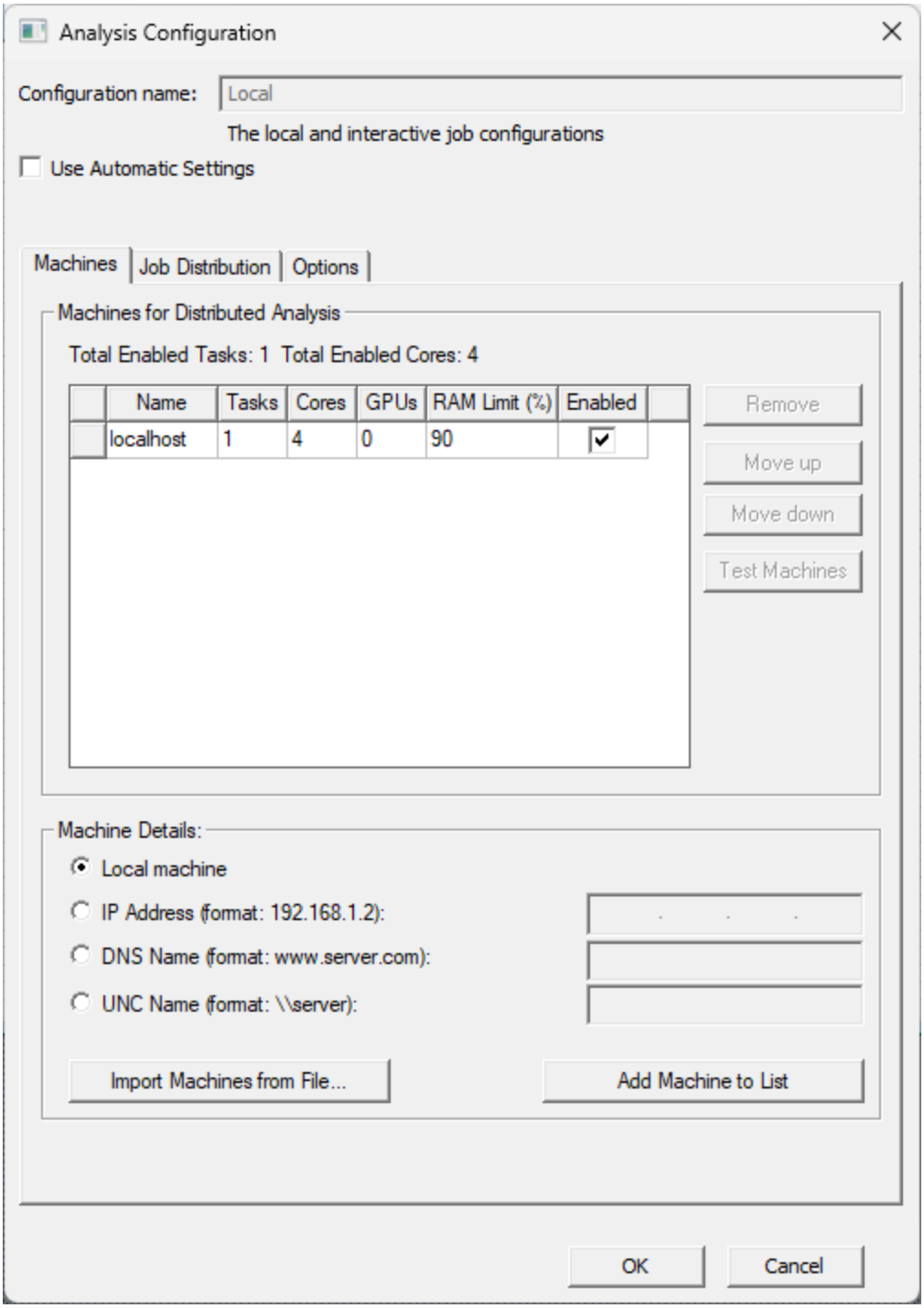
To create a new distributed machine configuration, or edit an existing machine configuration, for HFSS 3D Layout, see High Performance Computing: Distributed Analysis: Editing Distributed Machine Configurations.

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

Distributed Machine Configurations for Ansys SIwave™ in HFSS 3D Layout

To create a new distributed machine configuration, or edit an existing machine configuration, click **Tools** > **Edit Active Analysis Configuration** to open the **Analysis Configuration** window directly or click the **Analysis** settings icon on the toolbar.

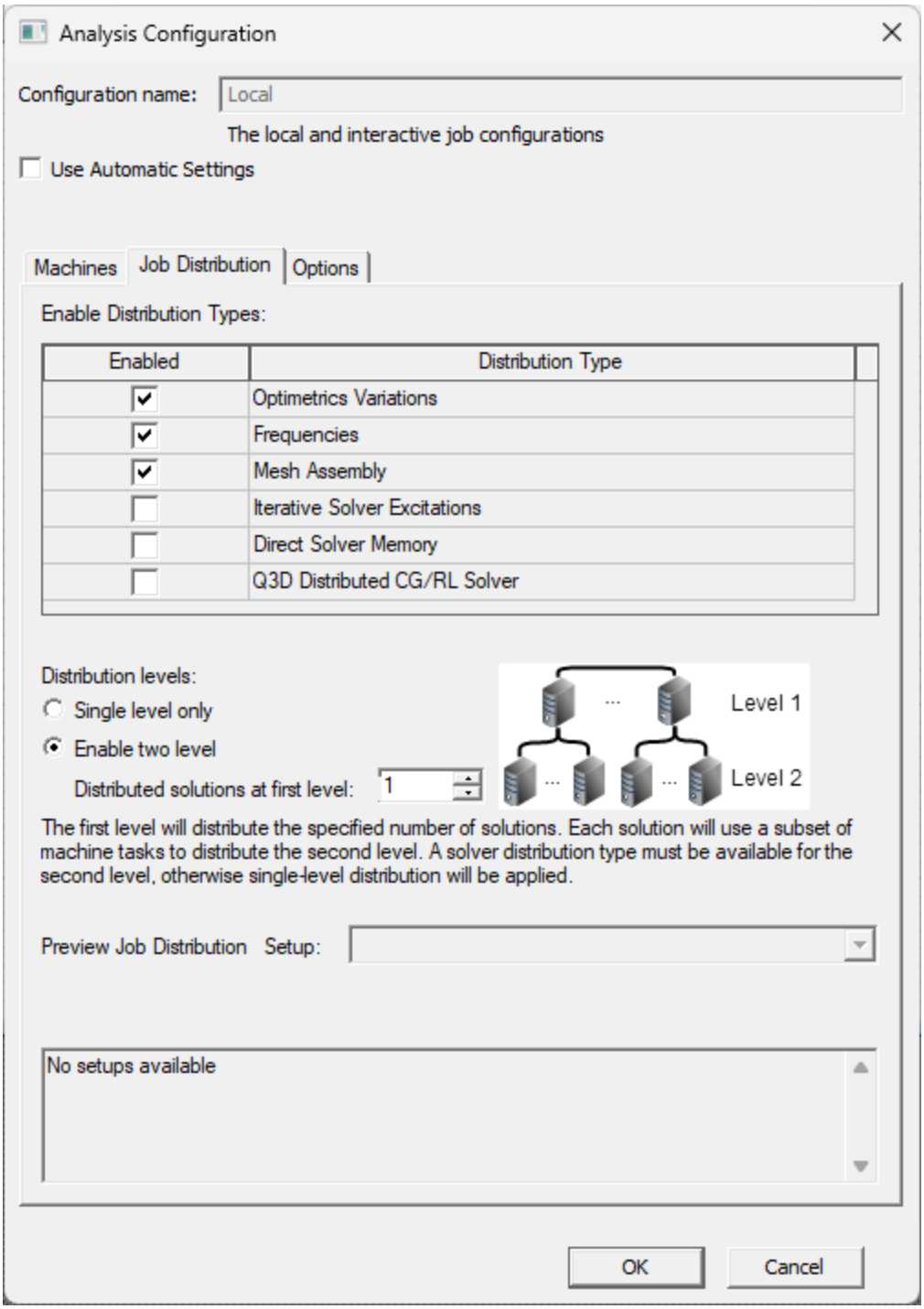
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Job Distribution Tab

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

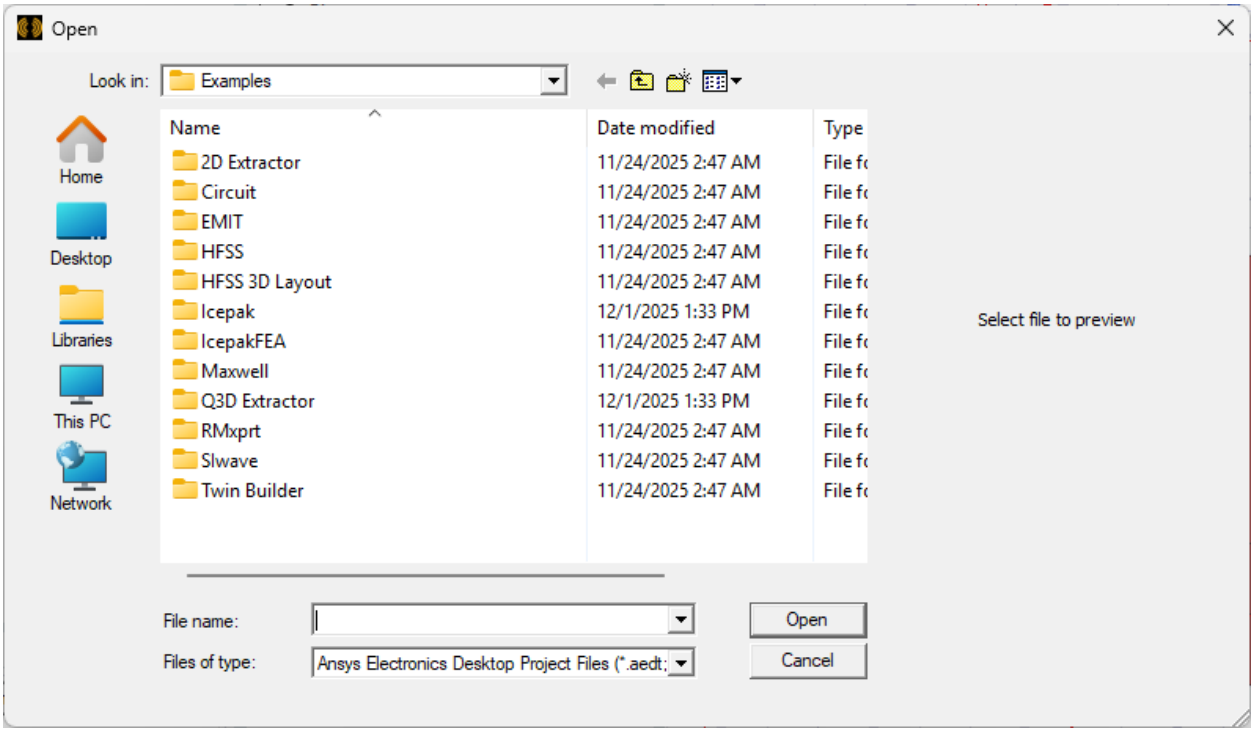
When Level 2 is enabled for Ansys SIwave™ printed circuit board (PCB) and package electromagnetics simulation software parametric setups, the first level distributes variations and the second level (SIwave solver) distributes frequencies.



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

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. The examples under Circuit, HFSS, and HFSS 3D Layout include subfolders, organizing the projects according to different applications:

- Circuit has subfolders for Automation, High Speed Digital, Low Noise Amplifier, RF Microwave, and Signal Integrity examples. The High Speed Digital folder has further subfolders for PCIe and USB projects. And the RF Microwave folder has further subfolders for Amplifiers, DC-IV, Filters, Misc, Mixers, Oscillators, and System projects.
- HFSS has subfolders for ADAS, Antennas, EMI EMC, Filters, Metamaterial, RCS, RF Microwave, Signal Integrity, and Transmission Lines.
- HFSS 3D Layout has subfolders for Antennas, Component, Filters, FSS, and Signal Integrity.

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

- HFSS 3D Layout Example Projects

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

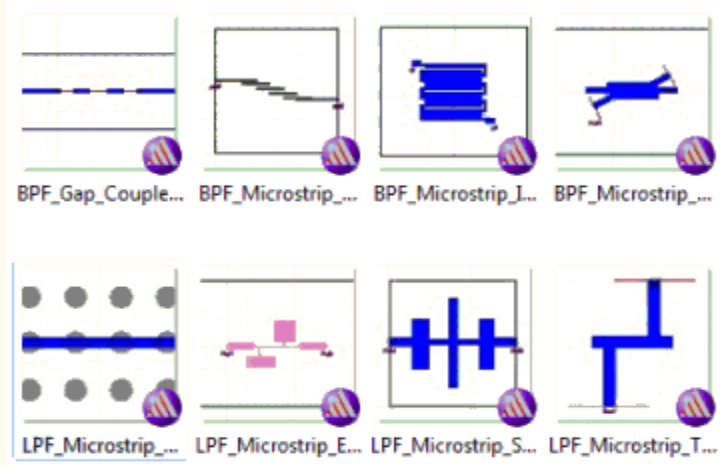
HFSS 3D Layout Example Projects

The examples directory for HFSS 3D Layout has subdirectories for Antennas, Component, Filters, FSS and Signal Integrity.

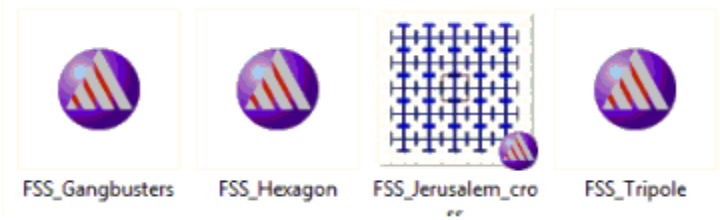
- Antennas

- Components

- Filters

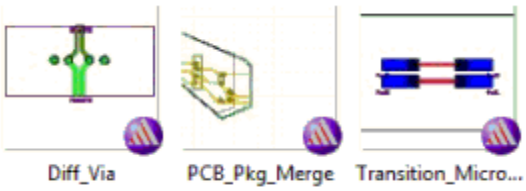


- FSS



- Signal Integrity

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



Finding Information in the Online Help

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

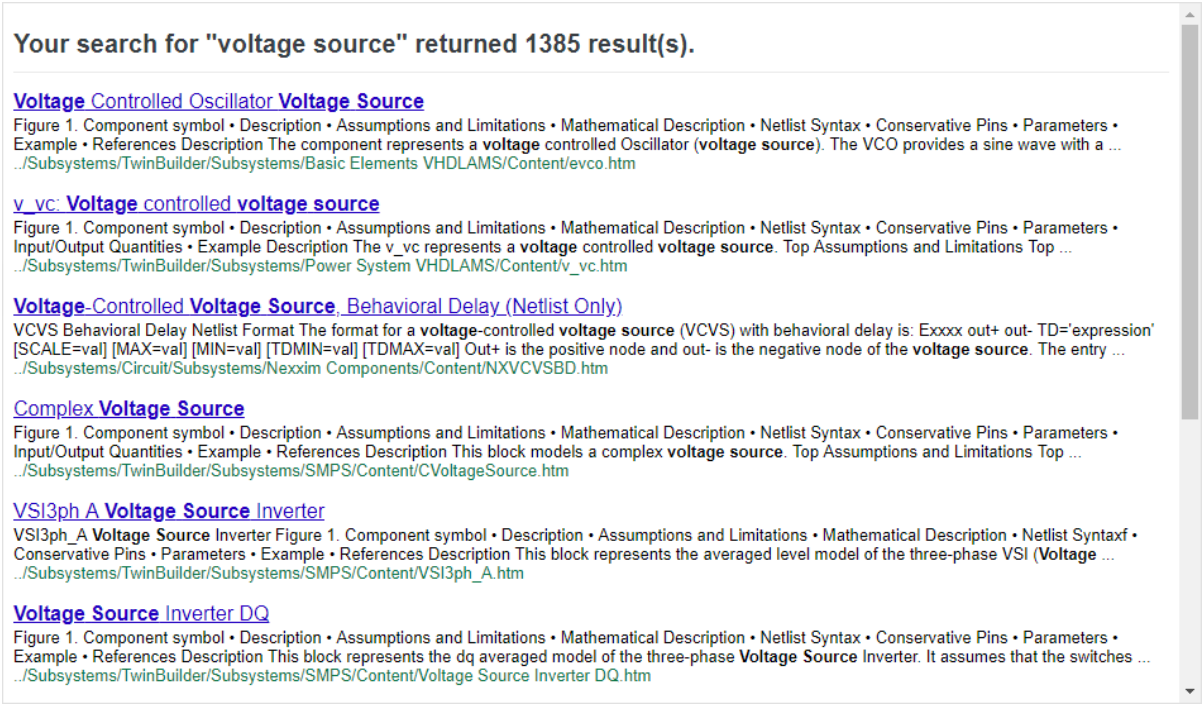
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase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

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

Using the Search Function in the Help

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

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

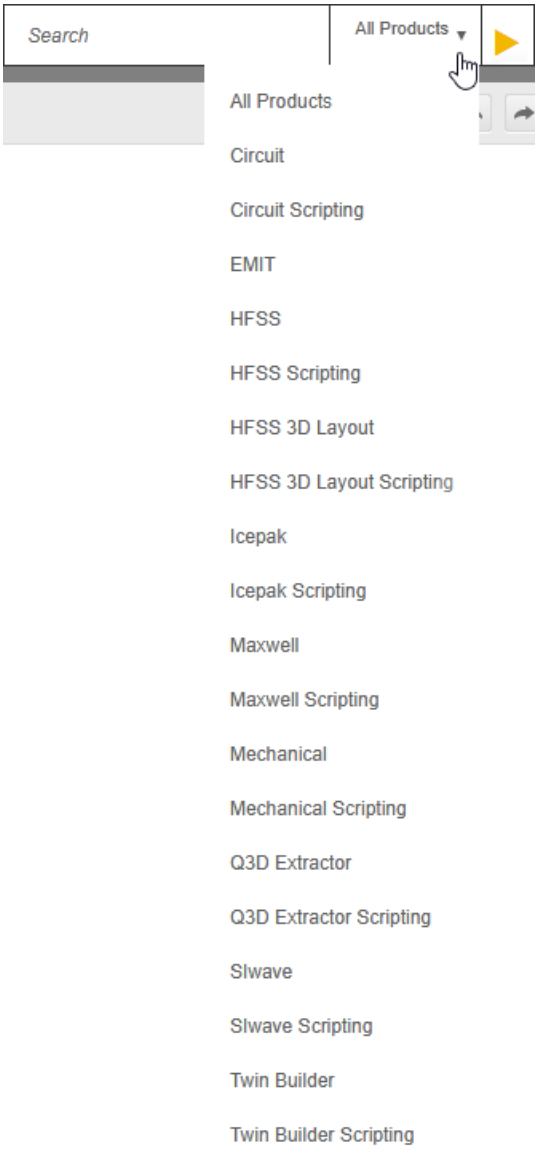


This method probably provides more hits than you want. The search function in the help provides several methods for making searches more specific.

Performing a Basic Search

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

Ansys Electromagnetics Suite 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.

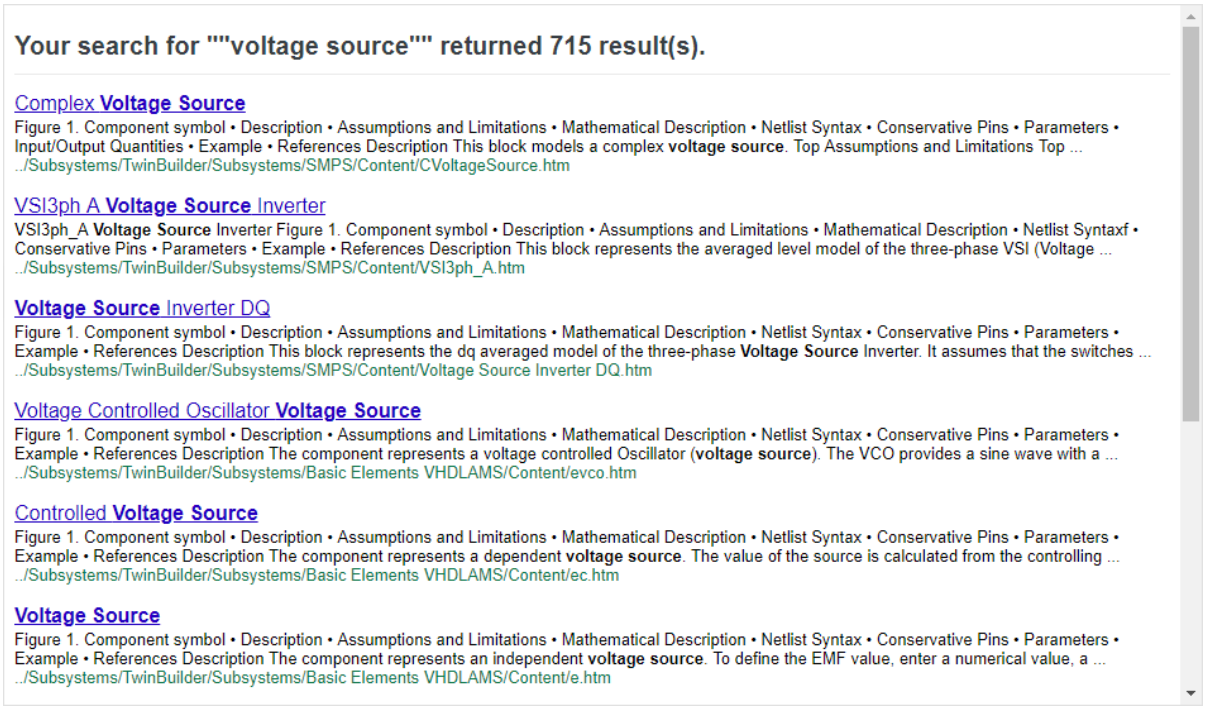
2. Click on the topic you want in the results list.
   - Some topics in different products share the same title. When searching "All Products," check the URL below each link. The URL indicates the product under which the topic falls.
   - After clicking a topic, if you want to view a different topic,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

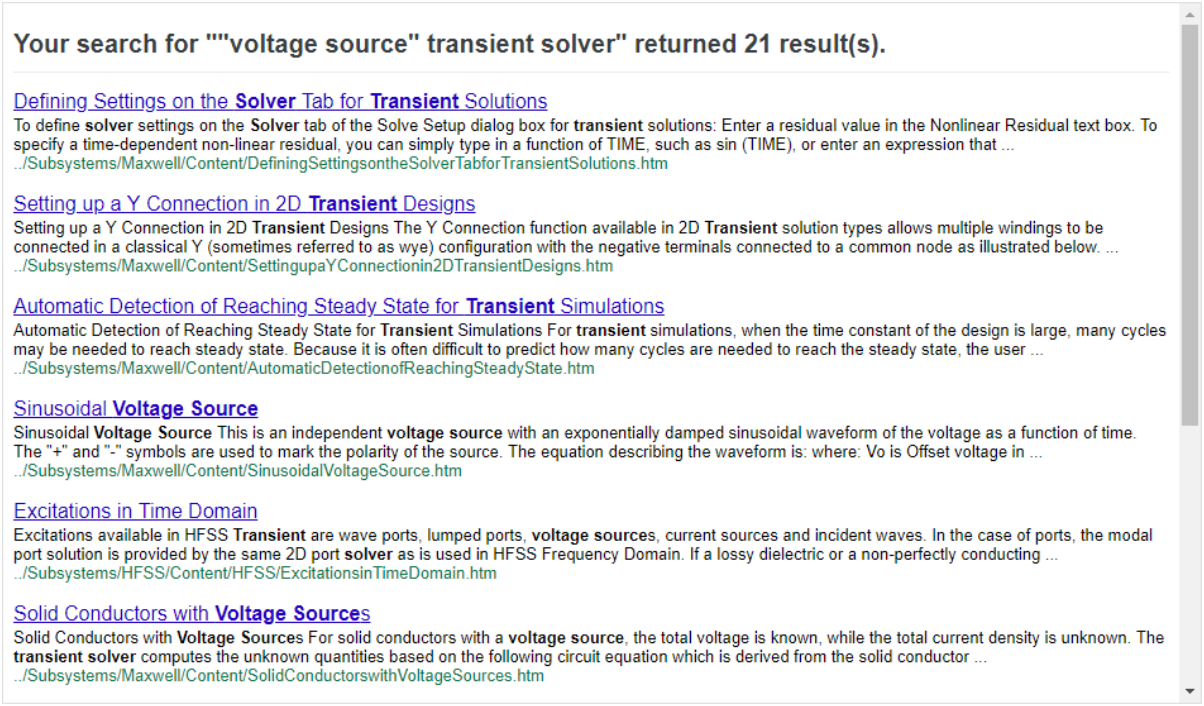
If you enter “voltage source” with quotation marks, the results show all topics that include the phrase.



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**.

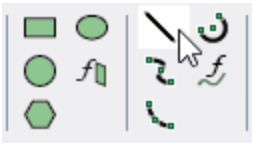
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

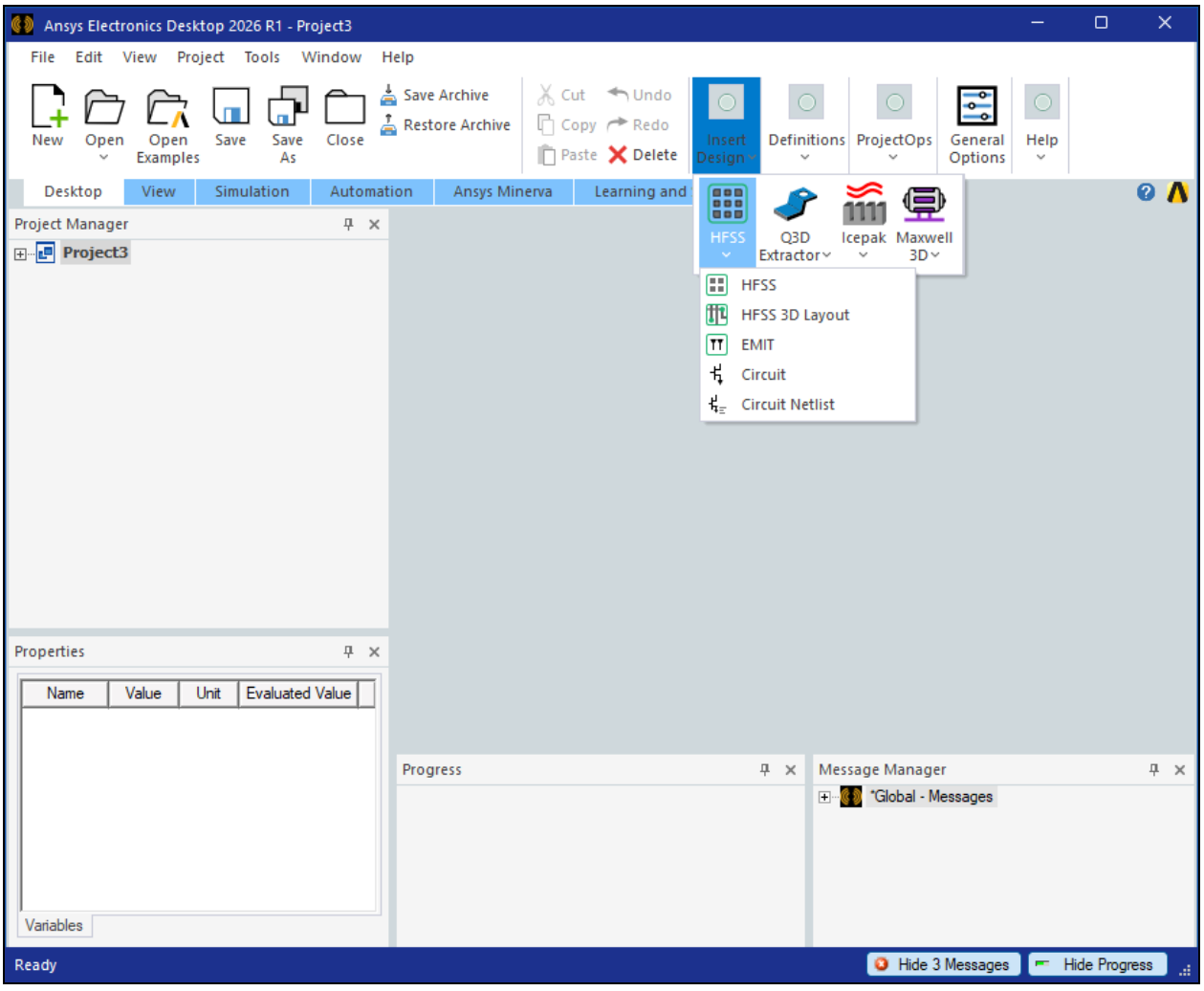
E-mail can work well for technical support. All Ansys EM software files are ASCII text and can be sent conveniently by email. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached. This allows more rapid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

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.
- **Maxwell 2D** – 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.

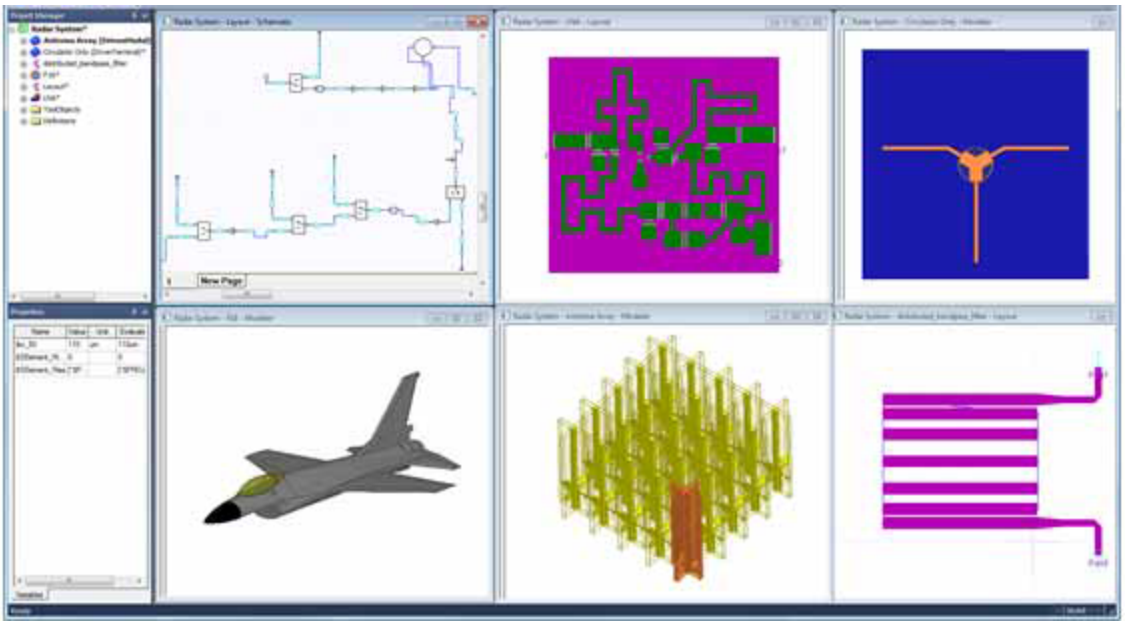
You can access all of these design types and features from the **Project** menu, and any combination of design types can be inserted into a single project file. The schematics can be used to wire up the different field solver models and create a model of a high-level system. Ansys Electronics Desktop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

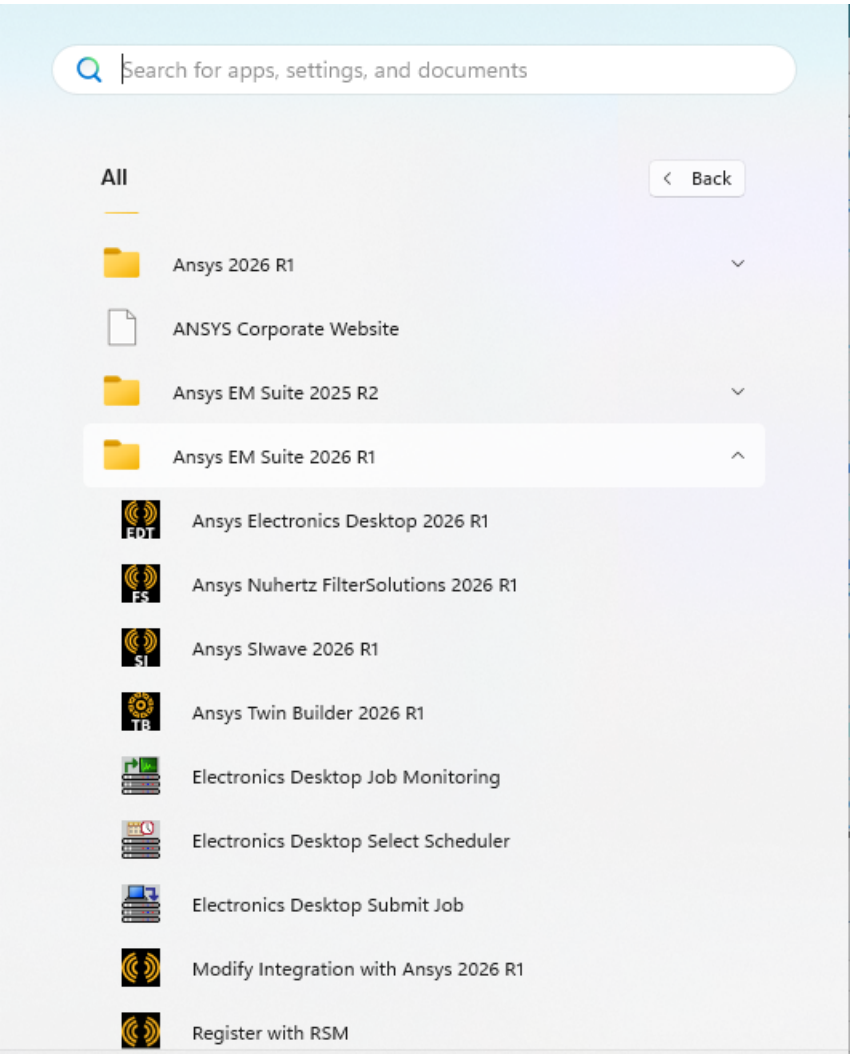
Follow the instructions below to launch Ansys Electronics Desktop in either Windows or Linux.

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?*

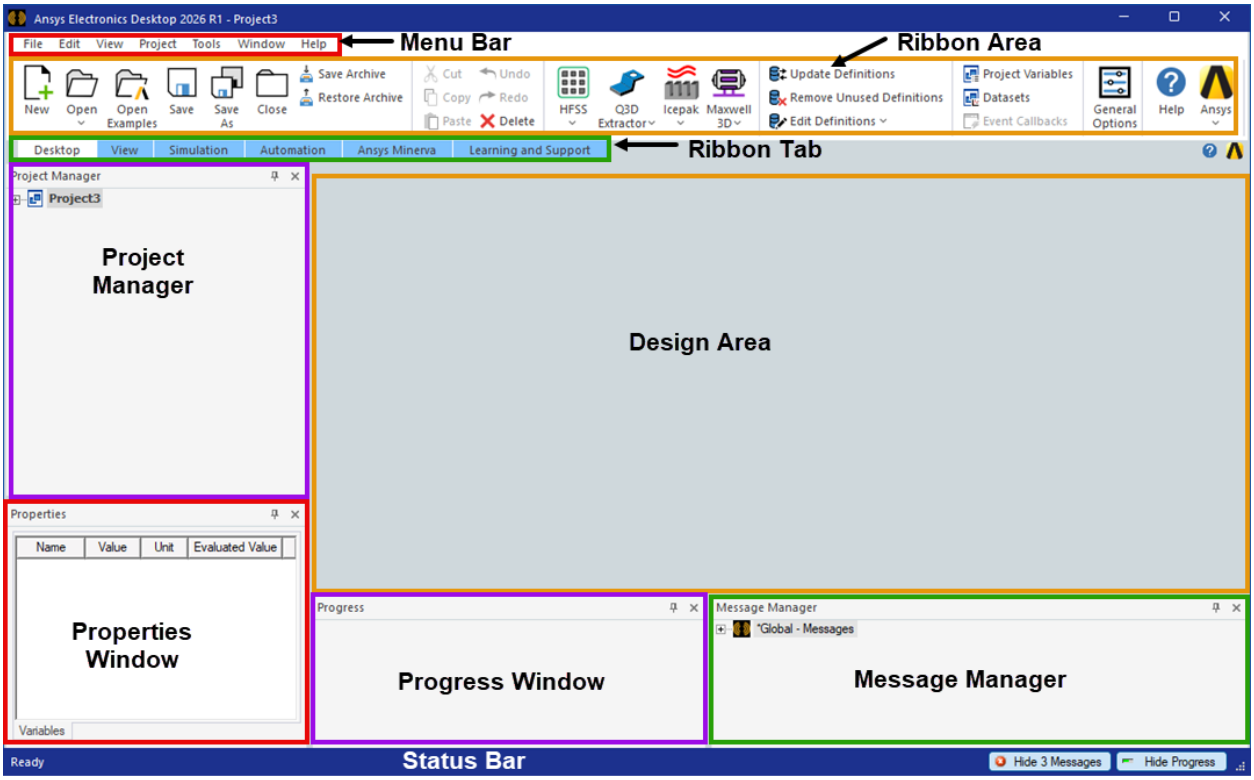
   In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

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.

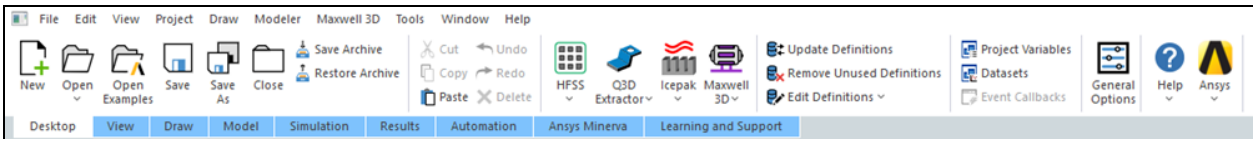
> **Note:**
>
> To ensure that all Electronics Desktop dialog boxes are visible on your monitor, it is recommended to not scale the display resolution above 100%.

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



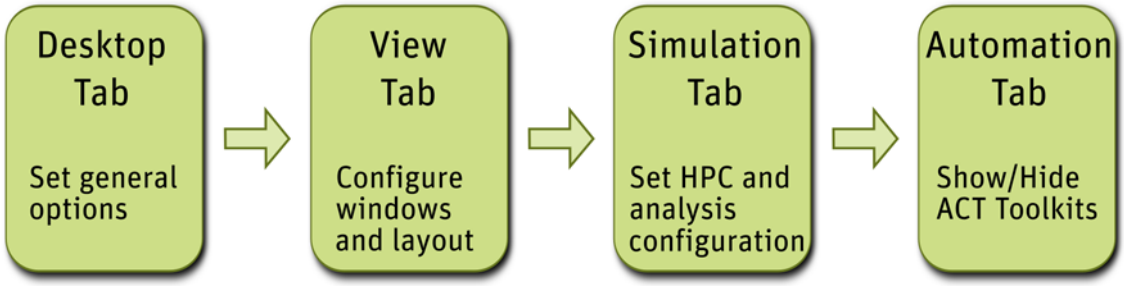
Working with Ribbons

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:

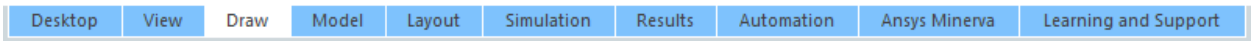
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

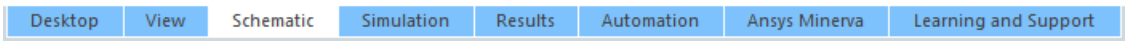
The **Desktop, View, Simulation, Automation, Minerva** and **Learning and Support** tabs appear for all design types. Additional tabs appear as follows:

- 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:

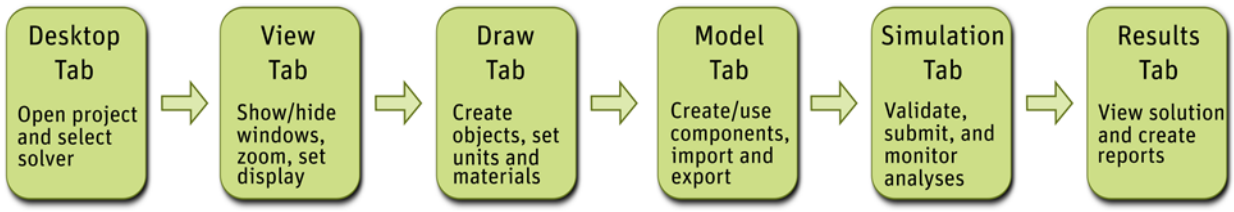


In addition to **Desktop**, **View**, **Minerva** and **Learning and Support** the ribbon tabs that are applicable to HFSS 3D Layout designs are described as follows:

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

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

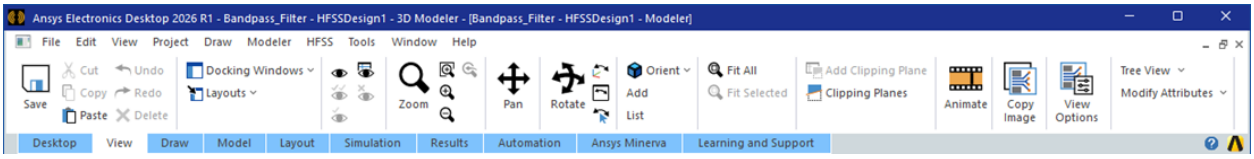
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



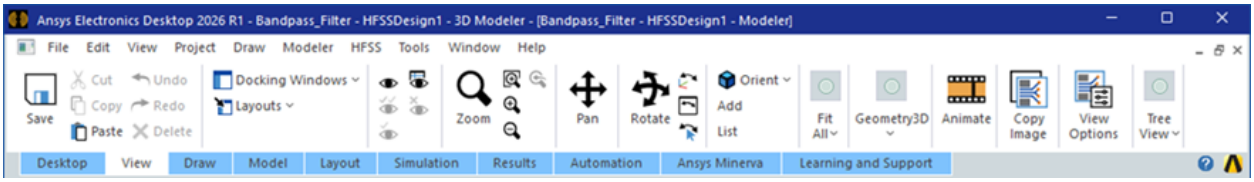
Note that workflow can vary depending on tasks at hand. For example, if you open a completed model in HFSS or use 3D components or the ACT Antenna Design toolkit (Windows only), you may not need the features of the **Draw** tab. If Validation on the **Simulation** tab identifies problems in a model,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



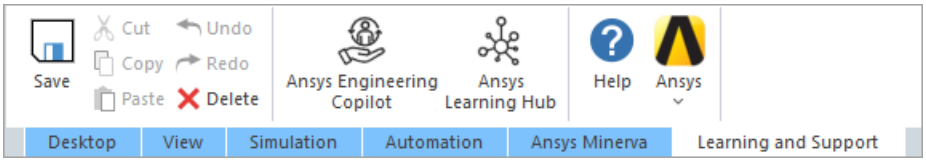
If you reduce the size of the Desktop window, the ribbon tabs become compressed. Fewer icons are shown, and available features are moved into drop-down menus rather than shown separately. The following example shows the **View** tab in a compressed state:



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

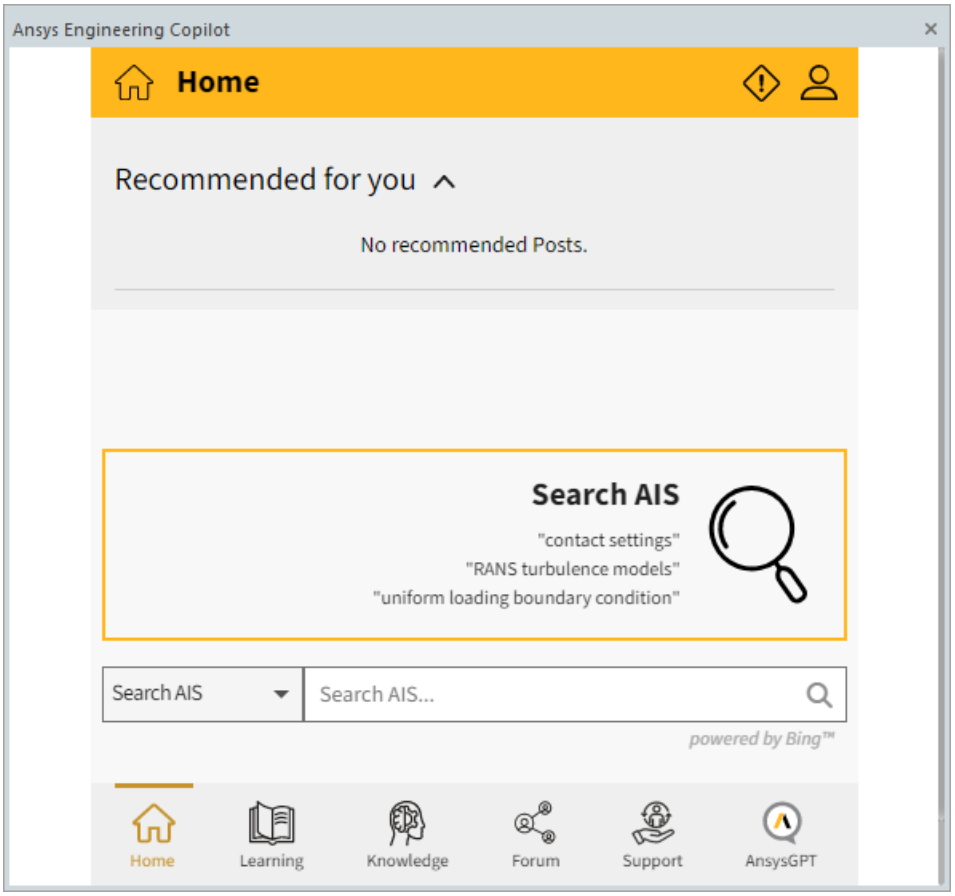
The **Learning and Support** tab provides easy access to technical advice, instruction, and examples on Ansys websites.

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:



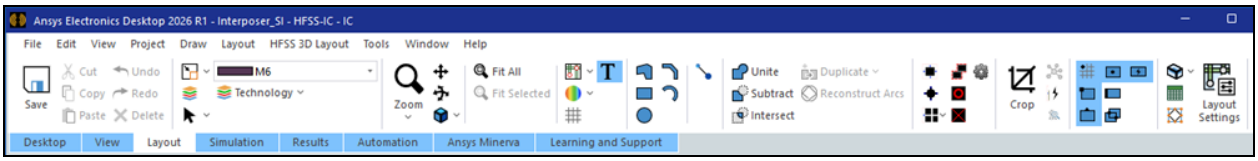
  - **Ansys Learning Courses** – Provide a wide range of Ansys Electronics Engineering courses using on-demand, self-paced video training and quizzes.
  - **Ansys Knowledge** – Provides expert curated knowledge materials from FAQs to tutorials on simulation topics.
  - **Ansys Forum** – Provides Ansys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

Ansys Electronics Desktop Windows

Ansys Electronics Desktop contains the following windows:

- Project Manager
- Message Manager
- Progress
- Properties
- Measure

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).

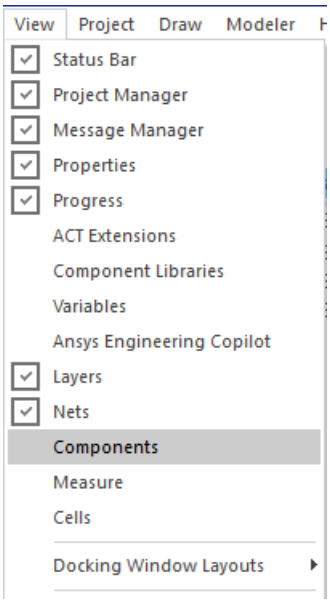
Showing and Hiding Windows

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

From the **View** menu:

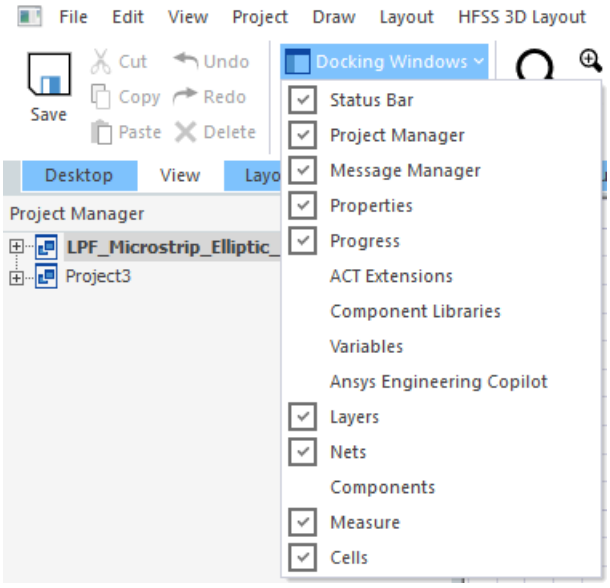
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click a window to show or hide it.



From the **View** tab:

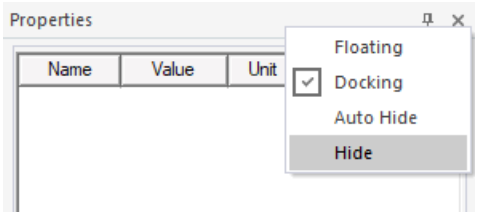
- Use the **Docking Windows** drop-down menu to show and hide windows.



You can also hide a window directly from 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.

- 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

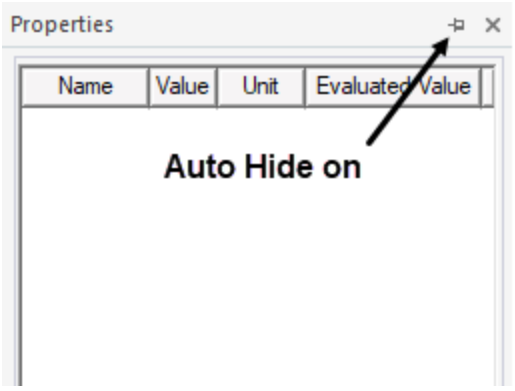
Windows can be moved out of the way so that only their title bars show. When you hover over this title bar, the window appears. This mode is called Auto Hide.

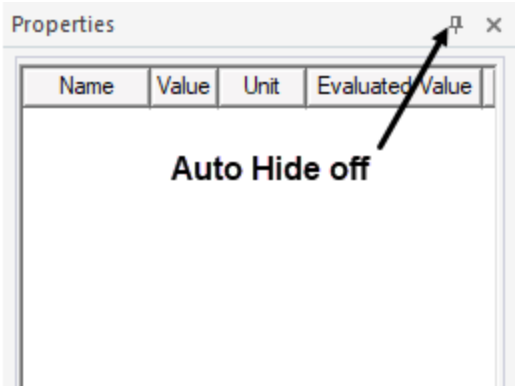
To automatically hide a window until the cursor is hovered over it:

- 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.

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

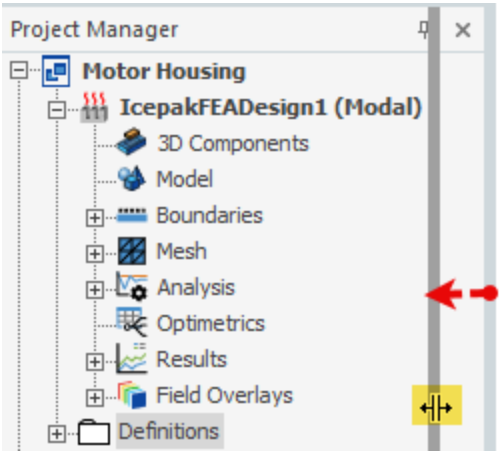




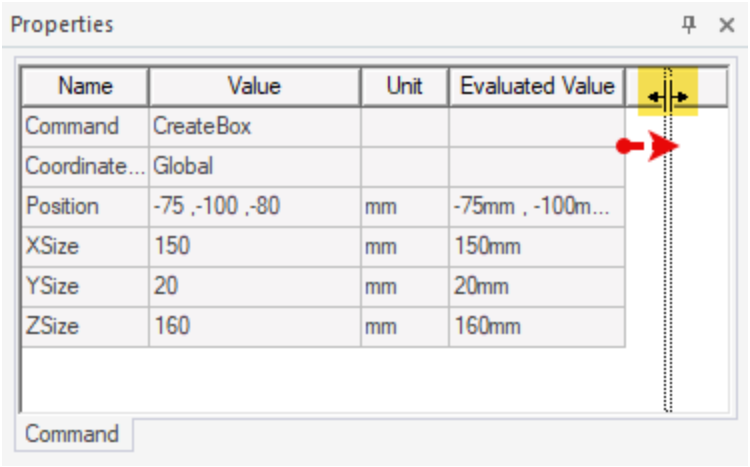
Moving and Resizing Windows

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):

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



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

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:

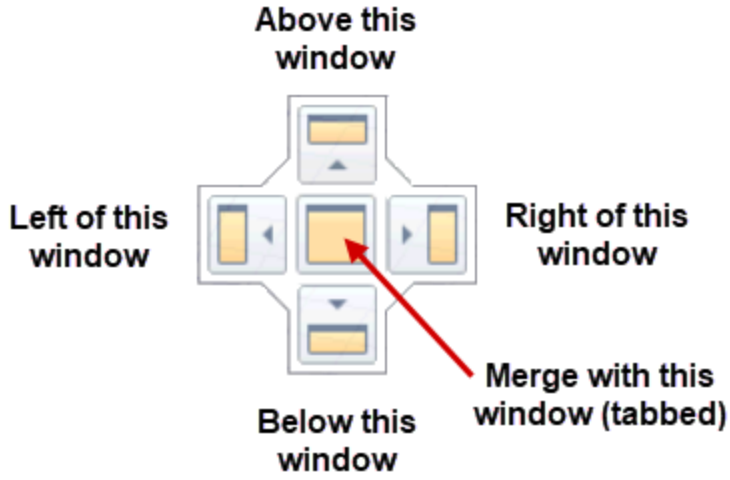
Right edge of user interface:

Top of user interface:

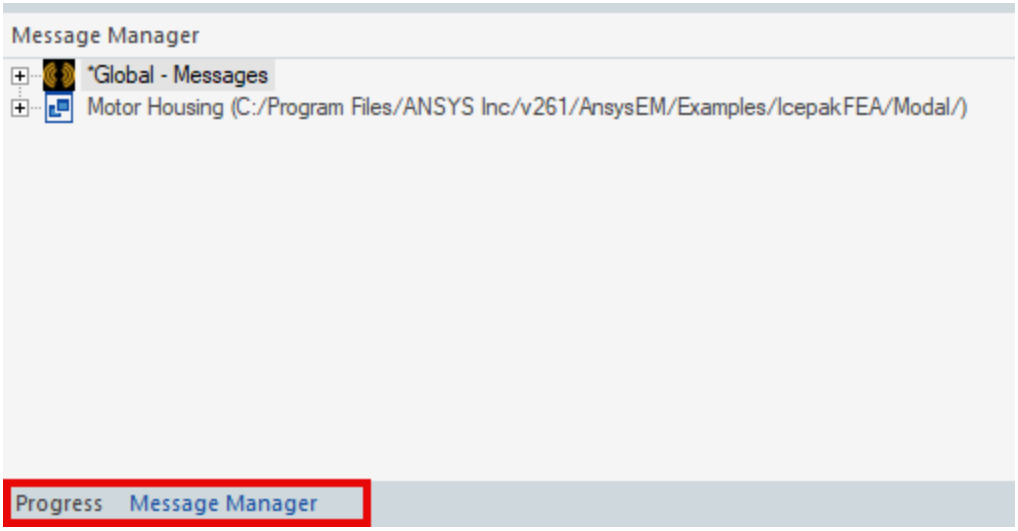
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bottom of user interface:

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



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



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

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.

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 Window

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

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

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

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

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

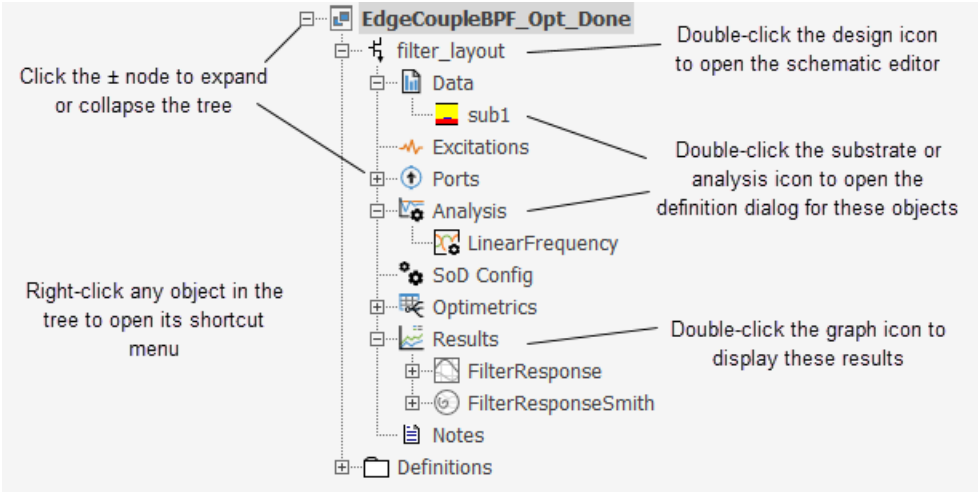
- Right-click in the toolbars area on the desktop, and then click **Project Manager** on the shortcut menu.

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

Project Manager Window for Circuit Designs

The Project Manager window of the Ansys Electronics Desktop for circuit designs shows the projects loaded into the desktop. Each project may consist of one or more product designs.

Each design's model data, ports, excitations, solution setup, and results are displayed as entries in a separate subtree on the **Project** tab in the **Project Manager** window of the Ansys Electronics Desktop.



To help facilitate the easy copying of design materials, Ansys Electronics Desktop allows you to drag and drop designs and components from the **Project Manager** window to the **Schematic Editor**.

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 drag and drop one or more copies of a component from the Components folder in the project tree to a schematic. Alternately, you can select Copy from the **Project** tree right-click menu and then select Paste from the schematic right-click menu (or use the Ctrl+C and Ctrl+V keyboard commands).
- If **Multiple Placement** is turned on for components in Ansys Electronics Desktop's Schematic Options dialog box, you can place multiple instances of a component in the schematic by clicking at multiple locations. To stop placing components during multiple placement, do either of the following:

    — Press **Enter**, the spacebar, **Backspace**, or **Esc** on your keyboard.

    — Right-click, and then select **Place and Finish**, **Finish**, **Cancel**, or **Back**.

Project Manager Tree Organization

Information contained in the design is organized into folders in the Project manager tree:

- **Data**—Substrate data, library references, netlist fragments, and so on
- **Excitations**—Sources in the circuit
- **Ports**—Ports in the circuit
- **Analysis**—Setup information to perform various analyses
- **Optimetrics**—Setup information to perform optimization, statistical analysis, and so on
- **Results**—Collection of graphs and tables for the design
- **Definitions**—The collection of components, symbols, footprints, and so on, accessible from Ansys Electronics Desktop

Double-clicking an item in these folders typically opens a dialog used to set the properties of that item.

Right-click Shortcut Menu

Right-clicking a folder or item pops up a shortcut menu that allows you to perform various operations. Virtually all project editing and management can be done from the **Project Manager** window using the right mouse button shortcut menus. Many of these menus are also available from the main menu bar in the various Ansys Electronics Desktop **product** menus.

Dockable Project Manager Window

The **Project Manager** window is a dockable window. It can be moved and sized any way you want it and it can attach itself to any edge of the Ansys Electronics Desktop.

Dragging and Dropping

To help facilitate the easy copying of design materials, Ansys Electronics Desktop allows you to drag and drop (or copy and paste) designs and components from 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.

to the **Schematic Editor**. In particular, in the **Project** tab of the **Project Manager** window, you can select and drag:

- A *design* from the Project Tree to a schematic
- A *component* from the Definitions folder to a schematic

You can also drag and drop (or copy and paste) designs in the Project Window onto a Circuit Design icon in the Project window.

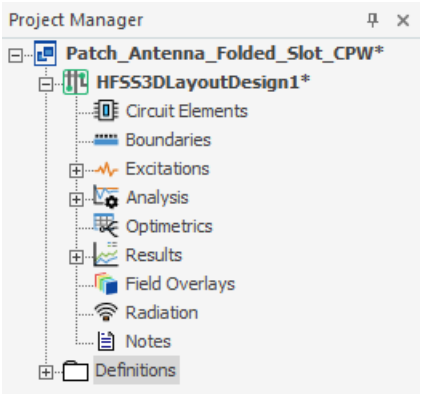
Dragging and dropping (or copying and pasting) an HFSS, Q3D, 2D Extractor, or SIwave design onto a Circuit Design or schematic creates a new dynamic link. Note If you hold the Ctrl key down while dropping or pasting, the appropriate Dynamic Link dialog will open, which allows you to make choices such as using an HFSS dynamic link in Transmission Line Model mode.

Closing the Project Manager Window

You can turn off the **Project Manager** window display by deselecting the **Project Manager** check box on the **View** menu in the Top Menu Bar. For more information, see the View Menu.

Working with the Project Tree

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



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

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

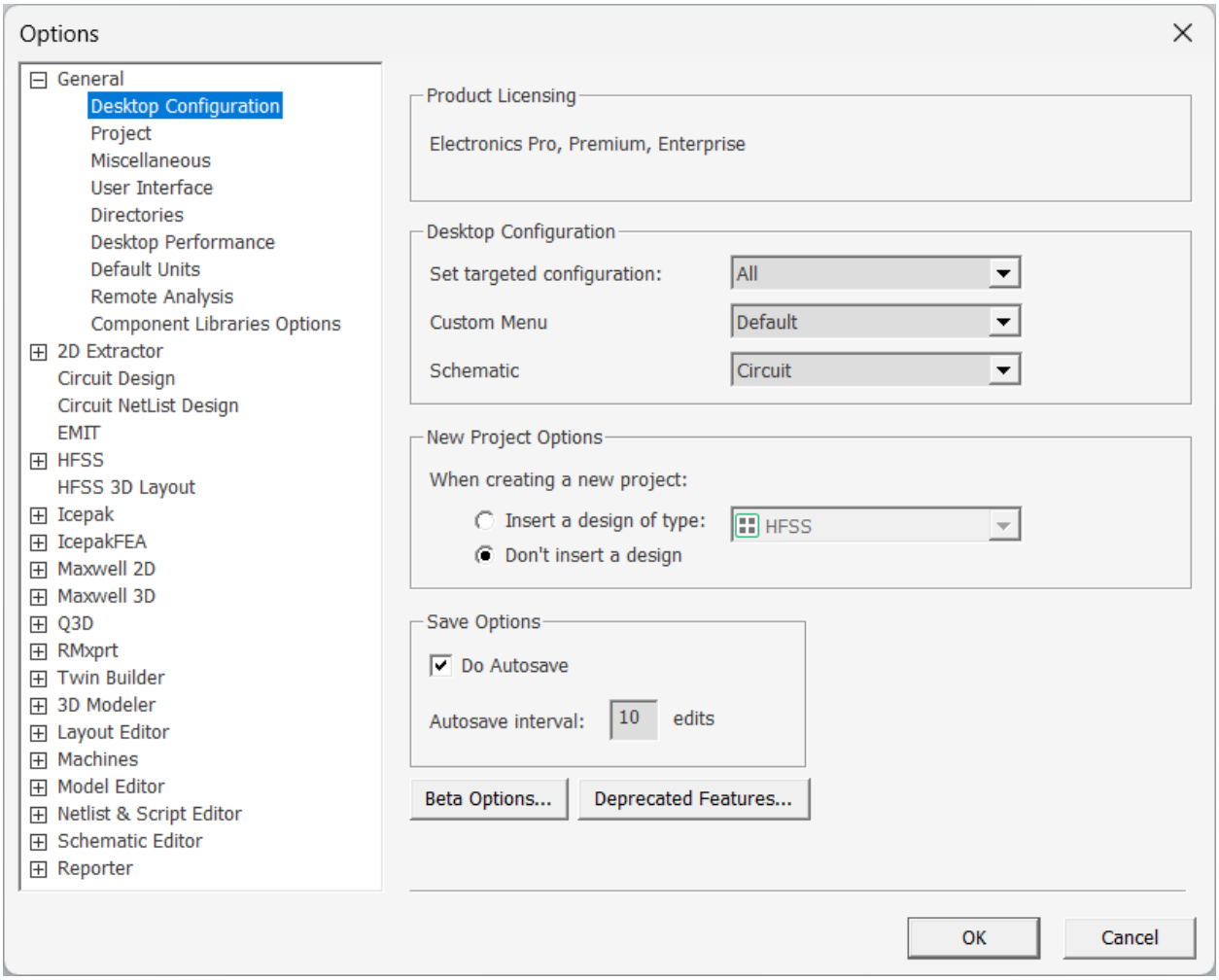
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the 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.

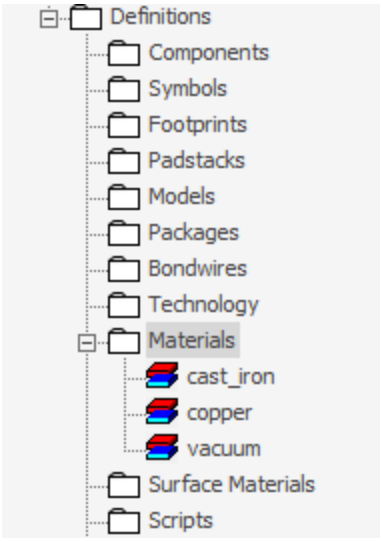


2. Click **User Interface**.
3. In the **Project Tree Visualization Options** area, check the **Expand Project Tree on Insert** check box.
4. Click **OK**.

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

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.



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.
Analysis	Displays the 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.
Results	Displays any that have been generated.

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

Field Overlays	Displays fields, which are representations of basic or derived field quantities on surfaces or objects. Plot folders are listed under **Field Overlays.** These folders store the project's plots and can be customized. See Setting Field Plot Defaults for information on how to customize the plot folders.
Documentation	Displays files you have added as documentation.
Definitions	Displays definitions for HFSS 3D Layout, 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.

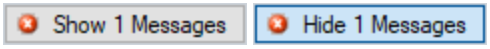
A window appears with that setup's parameters, which you can then edit.

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.

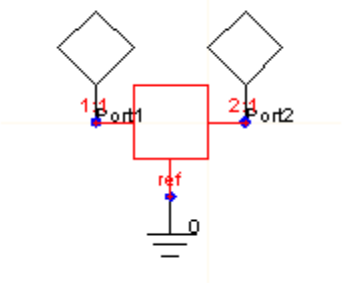
2. Expand **General** and click **User Interface**.
3. In the **General** area, check the **Ensure that new messages are visible in the Message Window Tree** check box.
4. Click **OK**.

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.

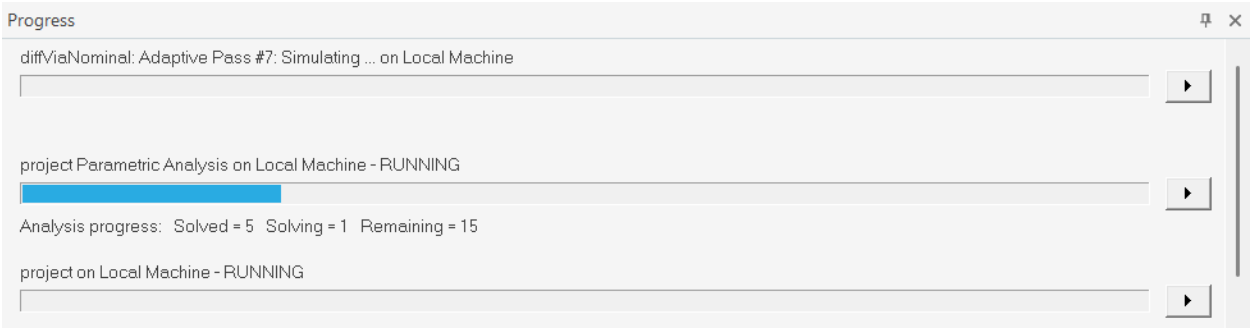
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Progress Window

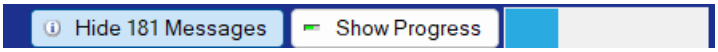
The **Progress** window monitors a simulation while it is running. Each simulation has its own progress bar. Right-clicking the bar allows you to abort the simulation or view simulation details.



To display or hide the **Progress** window, click **Show Progress** or **Hide Progress** on the status bar.

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

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

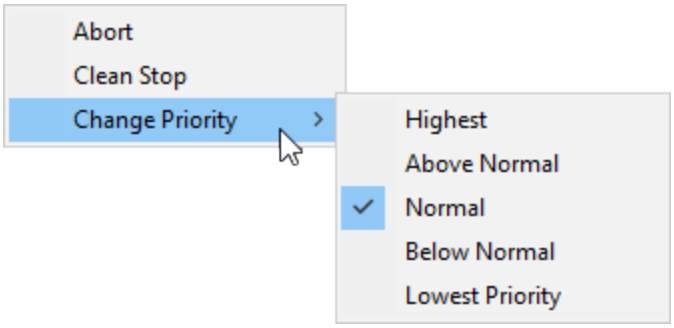


The progress window is also dockable, so you can reposition it.

Stopping or Aborting Simulation Progress

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

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

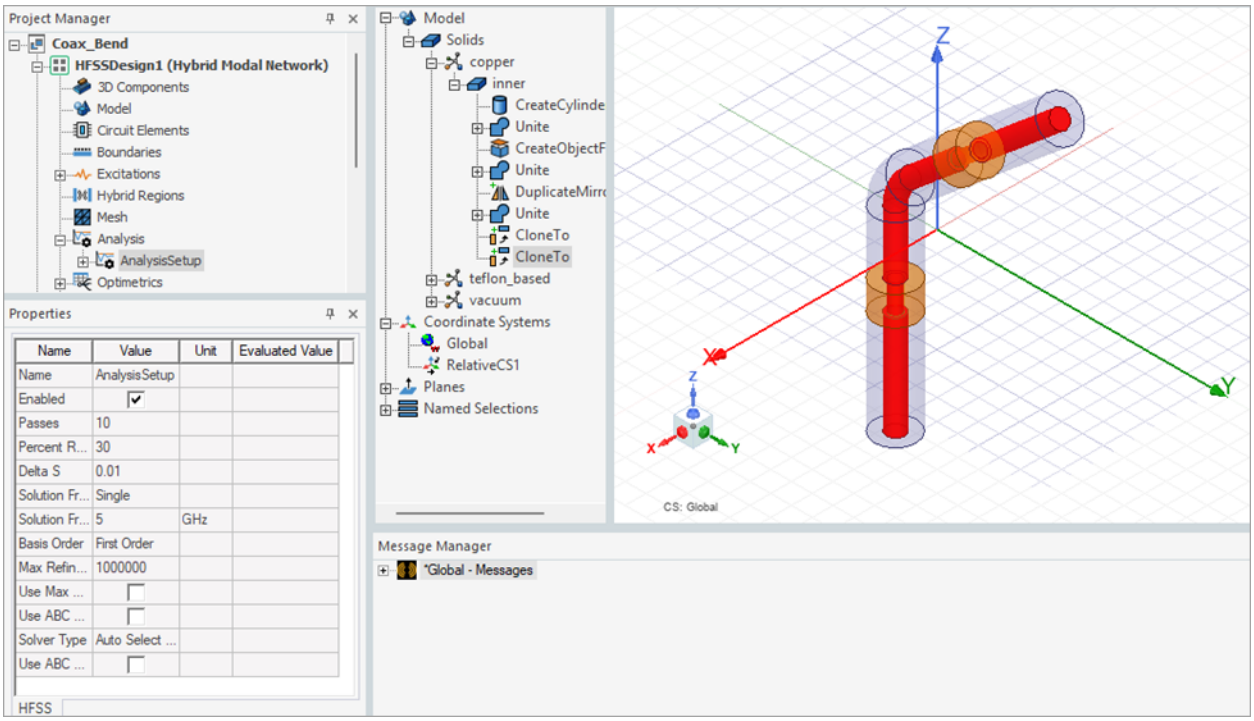


Properties Window

The **Properties** window displays and allows you to edit the properties of an item selected in the project tree 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.

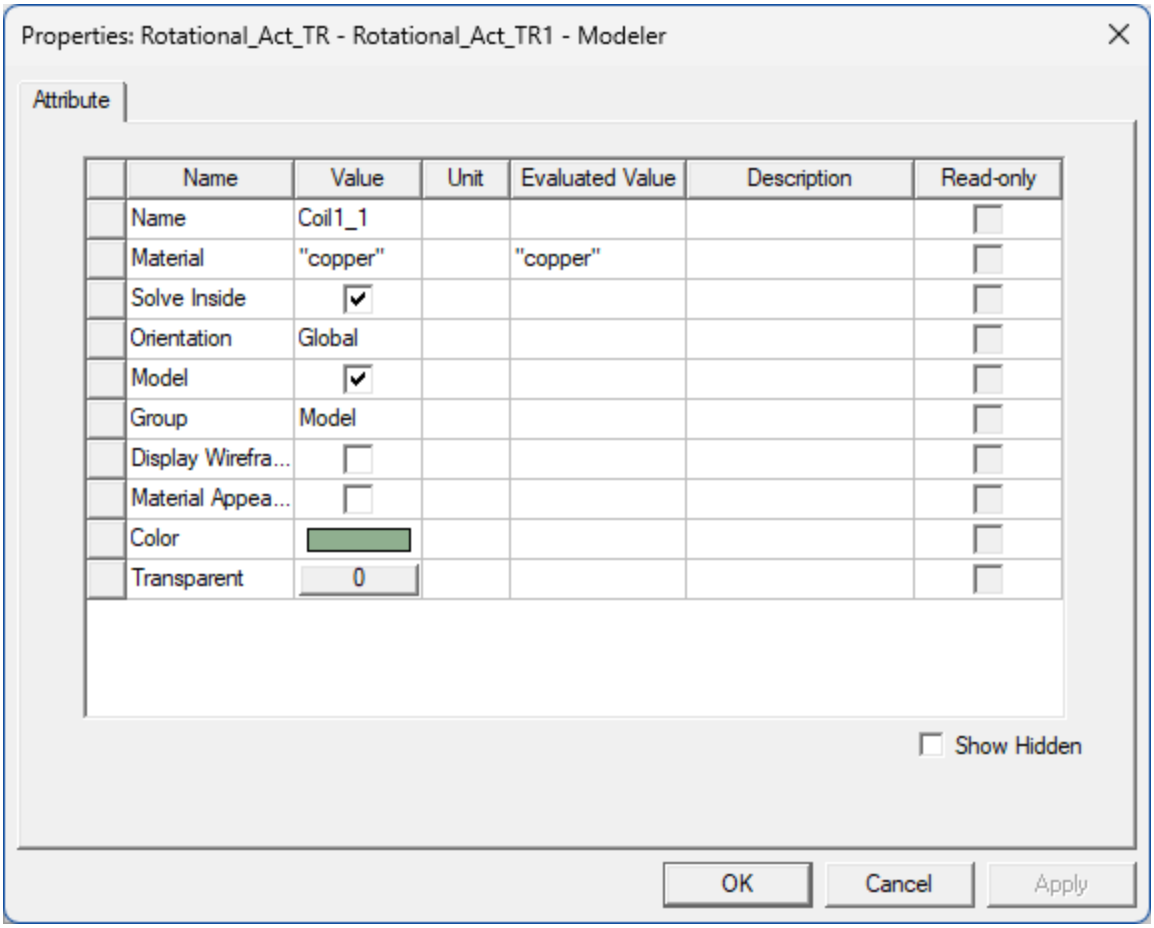
When the **Properties** window is docked, it displays the properties of any item you select in the Project tree, or the 3D Modeler window. Select **View** > **Properties** to remove the docked properties window.



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

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 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 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.



Opening the Properties Window

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

3. When you are finished making changes, click **OK**.

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

Setting the Properties 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.

Showing and Hiding the Properties Window

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

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

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

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

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

Modifying Object Attributes Using the Properties Window

1. Select the object for which you want to edit attributes by clicking it in the view window.
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.

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 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.

Properties Window for Schematic and Layout Editors

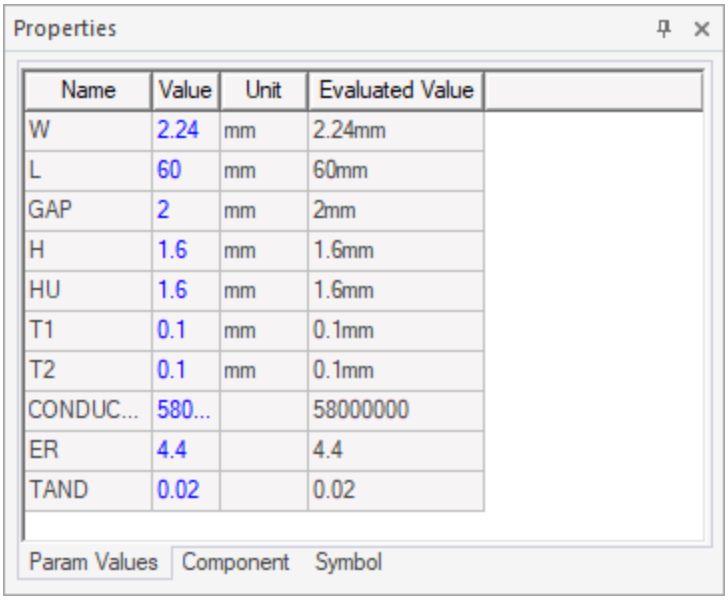
The **Properties** window shows the properties or parameters of objects selected in the schematic and layout editors, and objects in the **Project Manager** window. It allows you to modify property values. The **Properties** window has three tabs:

- The Param Values Tab

Param Values Tab

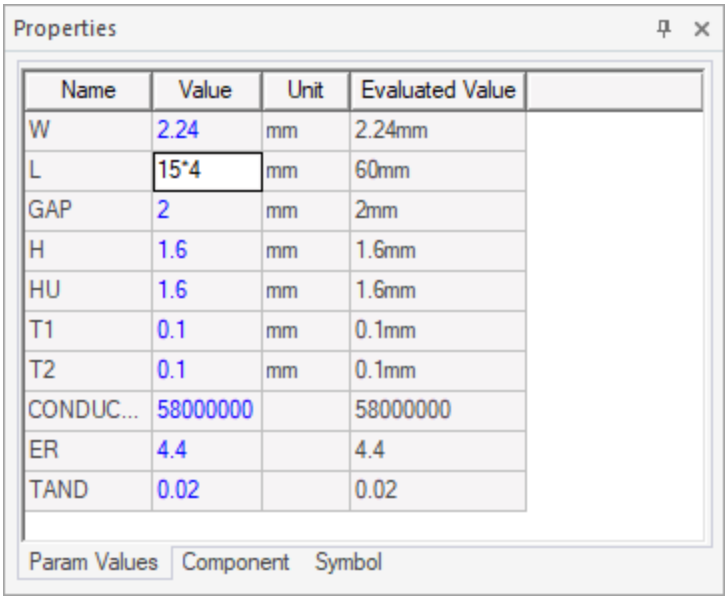
The **Param Values** tab in the **Properties** window of the Ansys Electronics Desktop lists the simulation parameters of the component or components selected.

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



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

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



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

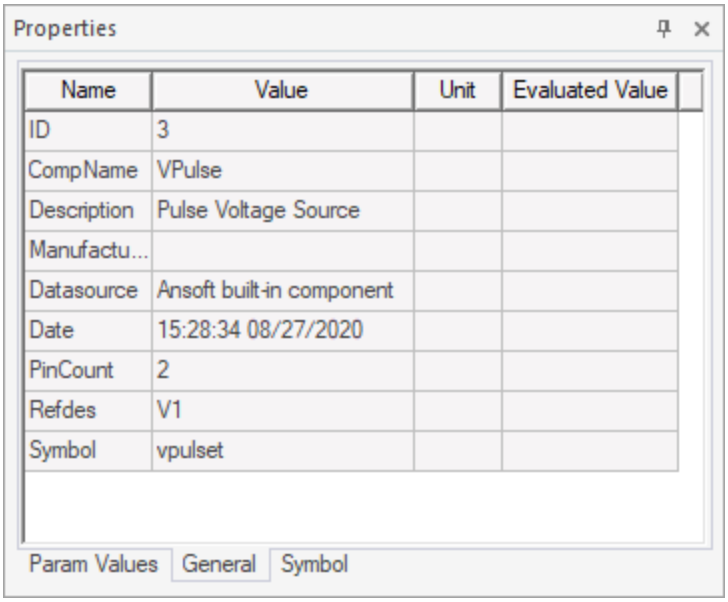
Launching Help from the Properties Window

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

1. Click in the **Components** tab of the Properties window.
2. Scroll down to the **Info** parameter entry, and then click the button in its **Value** cell.

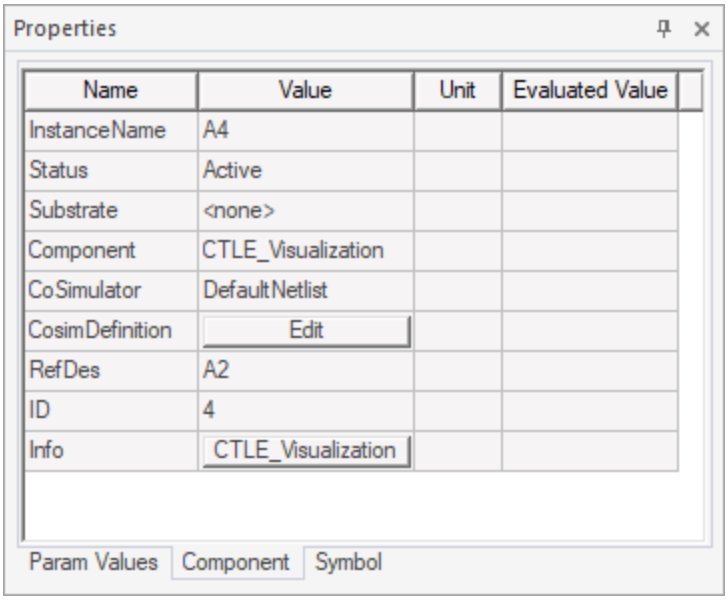
General and Component Tabs

If the selected component is from one of the component libraries, the Properties window will include the **General** tab, which lists the selection's name, symbol name, reference designator, and other data. These are generally not editable.



If the selected component has editable definitions, the Properties window will include a **Component** tab:

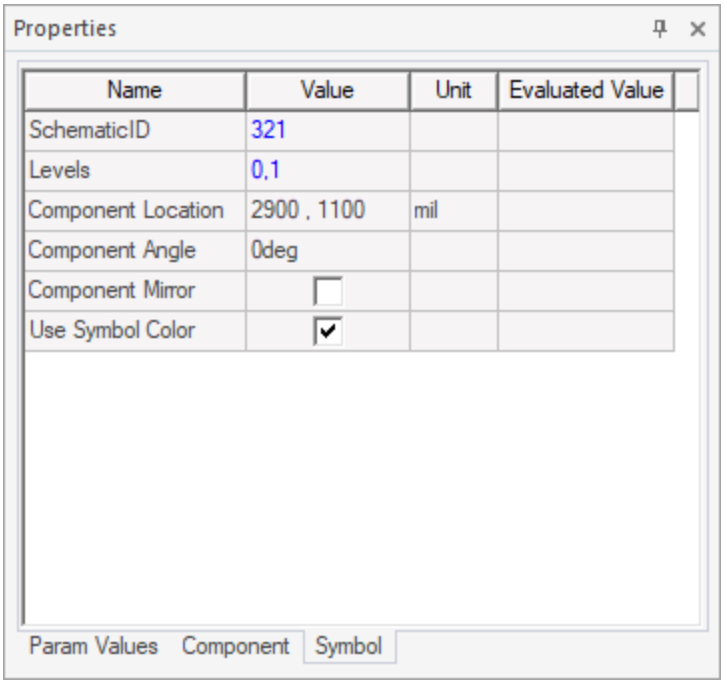
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This tab includes the instance name, its status, substrate layer, CoSimDefinition, and more. These fields can be edited.

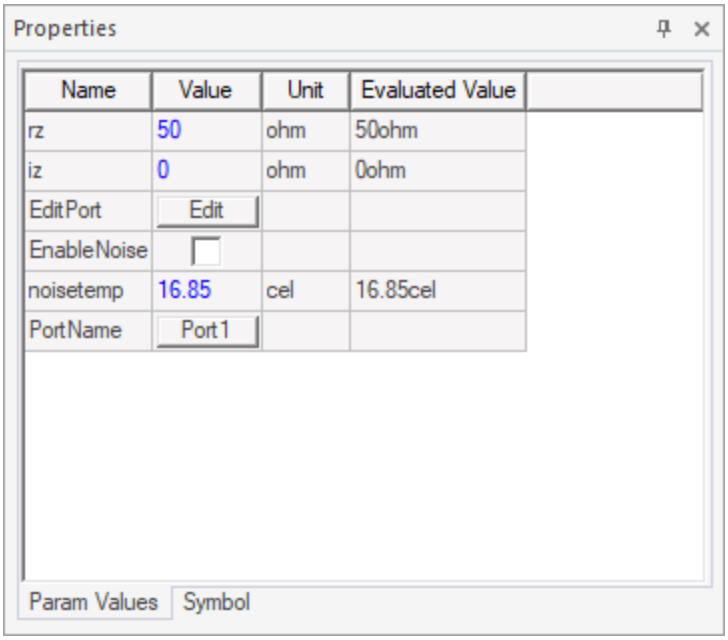
Symbol Tab

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



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

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



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

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

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

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

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 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:

- Component (displays if the circuit component definition can be edited)

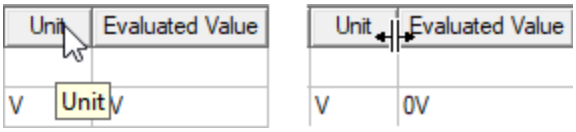
Opening the Properties Dialog Box

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:

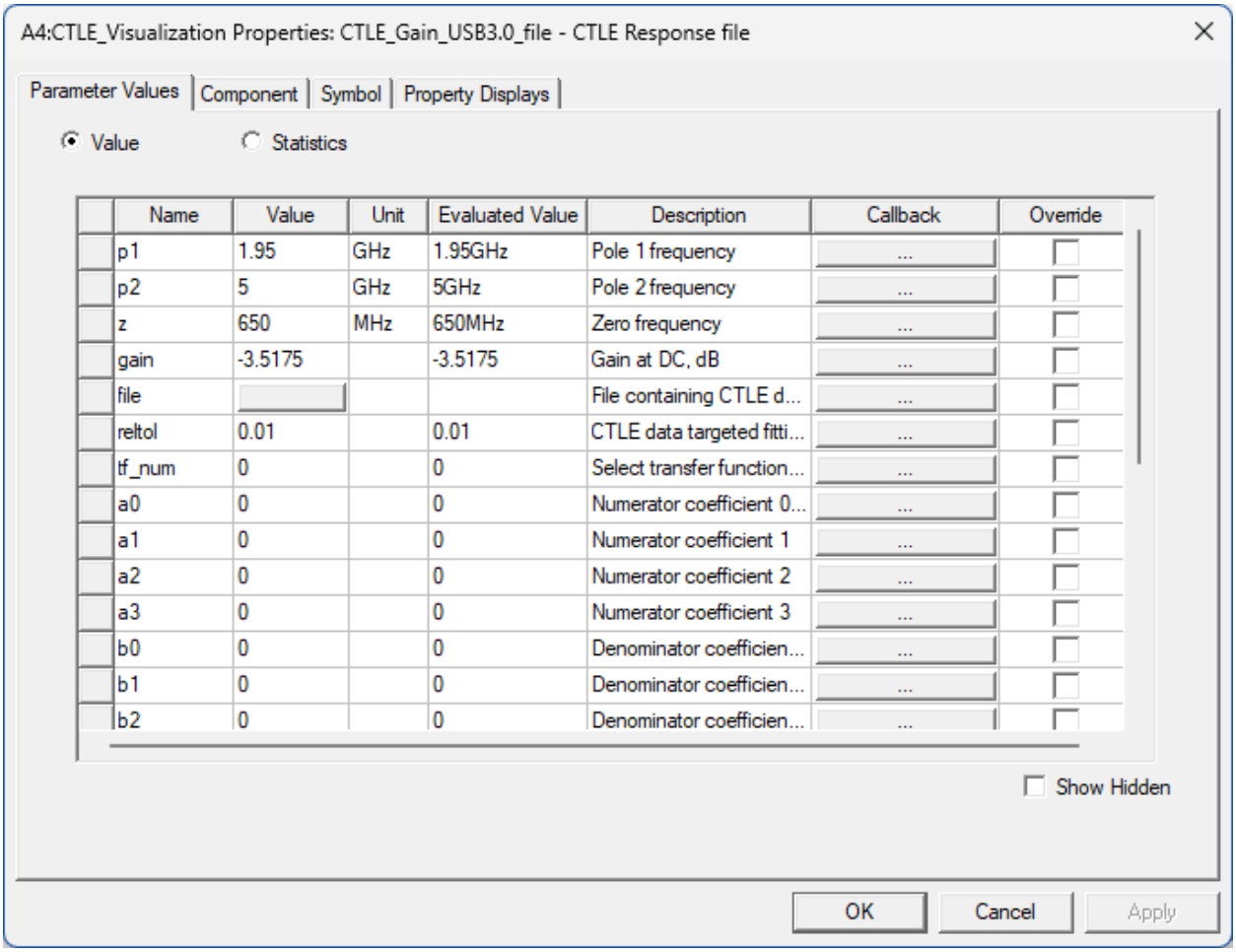
- Close the **Properties** dialog box by clicking the **OK** in the dialog box or by pressing the **Esc** key.
- Resize the **Properties** dialog box by dragging its edges.
- Change the relative widths of adjacent columns by dragging the header separators between them.



Parameter Values Tab

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

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



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

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

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

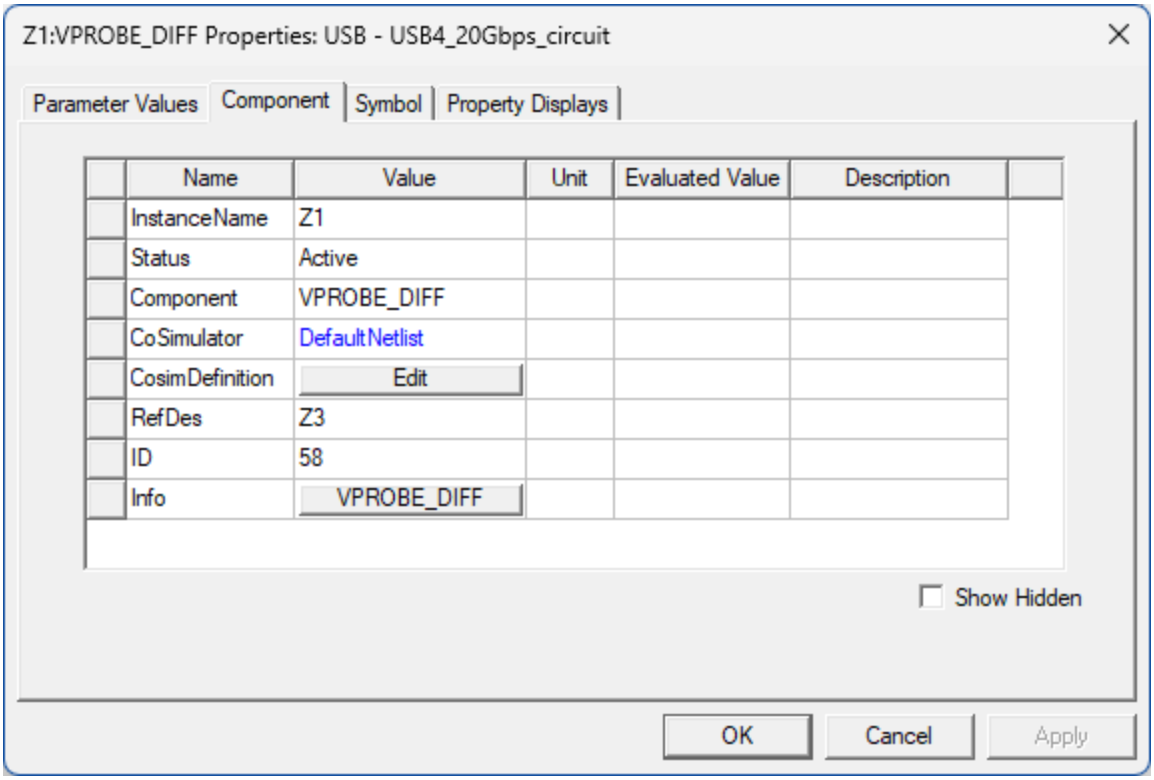
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

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

Component and Symbol Tabs

For designs with circuit components, the Component tab in the Properties Dialog Box is available if the selection's definition can be changed. It includes properties such as instance name, status, ComSim input, and substrate layer. Properties on this tab can be edited.

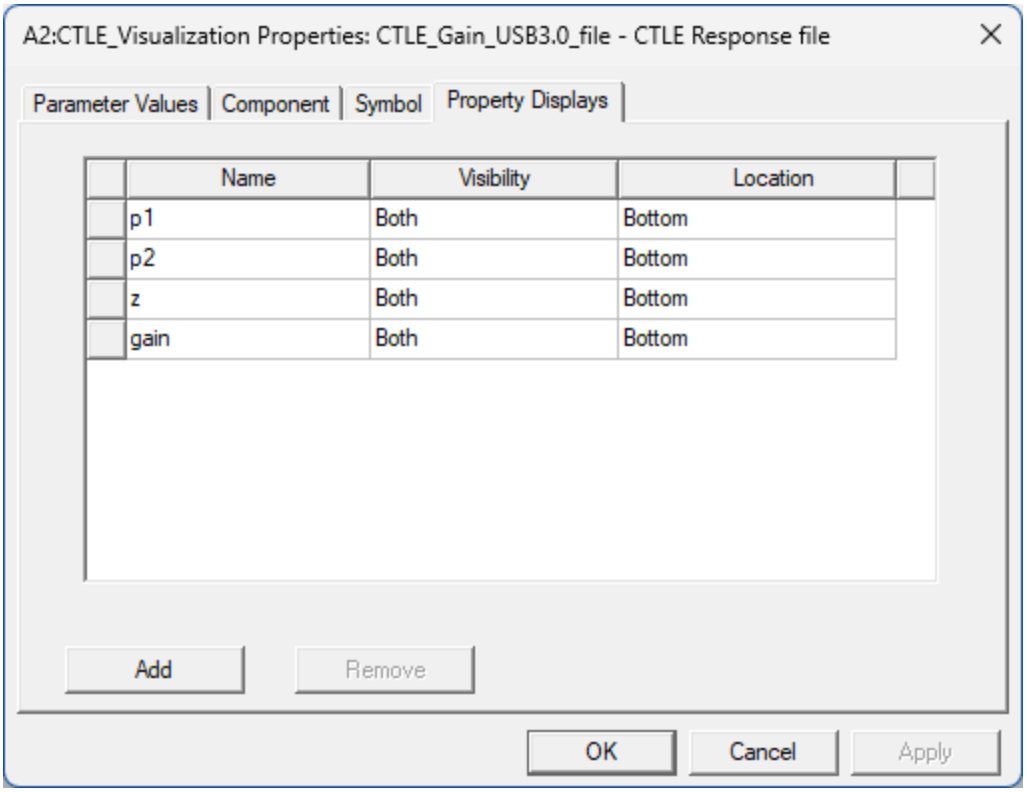


The **Symbol** tab provides information on the location of the component symbol in the schematic. The information is identical to that on the Symbol tab of the Properties Window.

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

Property Displays Tab

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



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

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

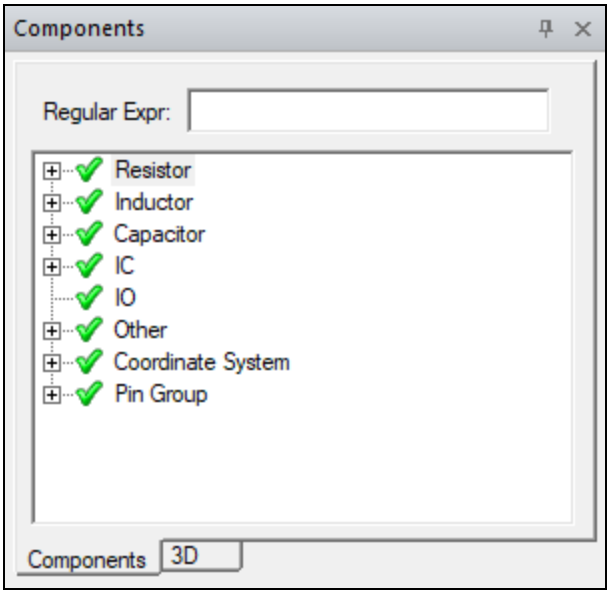
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Working with the Components Window

The **Components** window is a dockable window that is used to view and configure various component settings for Layout components. Use the **Components** window to select a component or a component class, add ports to a component, and configure multiple ports across components. You can also select multiple components in the Layout editor to configure pins and ports for each simultaneously. The window can be resized and relocated.



To show or hide the **Components** window on the desktop, click **View** > **Components**. A check box appears next to the command if the **Components** window is visible.

With a component selected, the following configuration controls are available from the right-click menu:

- **Type** – reclassifies the component type to **Resistor**, **Inductor**, **Capacitor**, **IC**, **IO**, or **Other**.
- **Enable**/**Disable** - enables or disables 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.

- **Model** – opens the Set Selected Component Models window, where you can change the component model definition, including die, solder ball, and port properties. Changing the model scope will update every reference designator that belongs to the model definition.
- **Create Ports on Component** – adds ports for the component
- **Remove Ports on Component** – removes ports for the component.
- **Fit Selected** – fits the selected component in the active view window.

> **Note:** When a component classification is selected, any changes are applied to all components within the class.

For more information, see Layout-based Component Encapsulation.

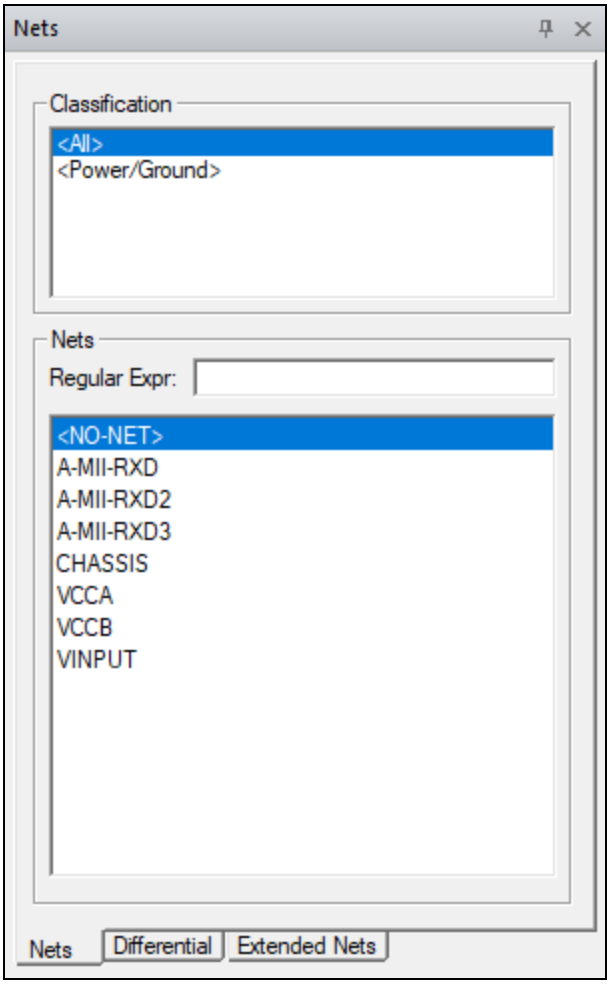
Working with the Nets Window

The **Nets** window is used to view and configure nets and collections of nets. It is a dockable window that can be resized and relocated. The **Nets** window is divided into three tabs:

- **Nets** – lists net and net classes.
- **Differential** – lists coupled positive and negative nets.
- **Extended Nets** – lists extended nets for use for SIwave SYZ with HFSS Regions simulations.

To show or hide the **Nets** window, click **View** > **Nets**. A check box appears next to this command if the **Nets** 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.



Nets Tab

The **Nets** tab contains two panes: an upper **Classification** pane and a lower **Nets** pane.

Classification Pane

A net classification (also called a class) is a collection of nets. Classifications are designed to group and organize nets by a common characteristic. A net can be in multiple classes at once. By default, there are two classifications: All and Power/Ground net class.

The **Classification** pane lists and manages net classes. When you select a class in the **Classification** pane, the nets assigned to that class are listed in the **Nets** pane.

The **Classification** pane's shortcut menu offers the following actions:

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

- **New** – opens the **Add Net Class** window for net class creation (see below).
- **Edit** – opens the **Net Class Properties** window for net class modification (see below).
- **Delete** – deletes the selected net class(es).
- **Select** – in the layout editor, selects the net classes selected in the classification pane.
- **Show** – in the layout editor, shows the net classes selected in the classification pane.
- **Show (Hide All Other)** – in the layout editor, shows the net classes selected in the classification pane and hides all other net classes.
- **Hide** – in the layout editor, hides the net classes selected in the classification pane.
- **Create Ports** – creates ports on all nets in the selected net classes.
- **Remove Ports** – removes ports from all nets in the selected net classes.

Nets Pane

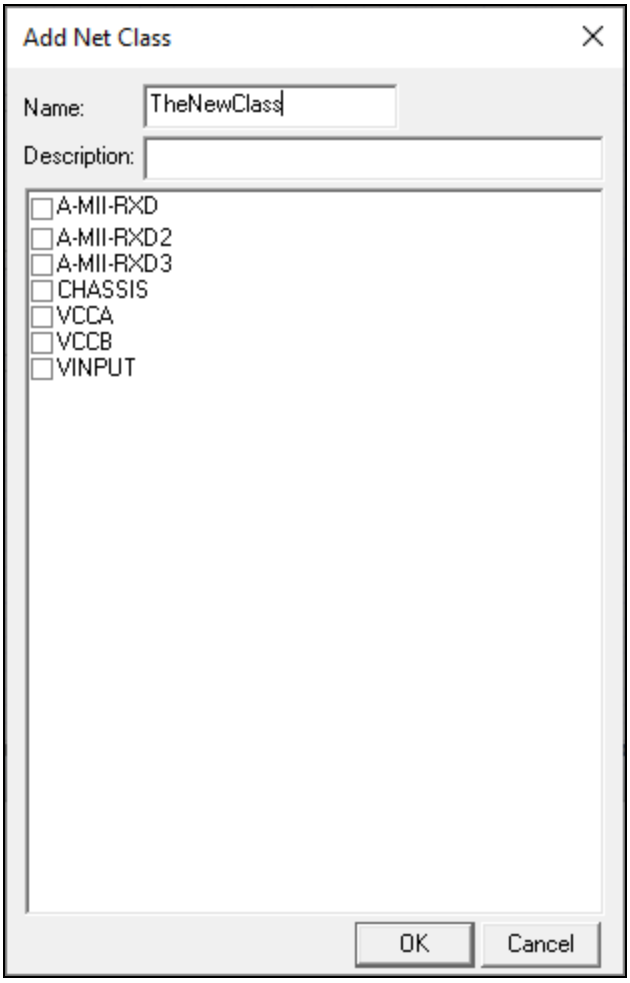
The displayed nets may be filtered using Perl Regular Expression syntax. In the **Nets** pane, the right-click menu contains additional options:

- **Add to Power/Ground** – adds one or more selected nets to Power/Ground.
- **Delete** – deletes the selected net(s).
- **Lists** – opens a list of all nets in the Design List window.
- **Remove from Power/Ground** – removes one or more selected nets from Power/Ground.
- **Create Differential Pair** – with two nets selected, groups them as a differential pair (see below).

Add Net Class

Selecting **New** from the right-click menu opens the **Add Net Class** window. Enter a name and description (optional) for the new net class and click **OK**.

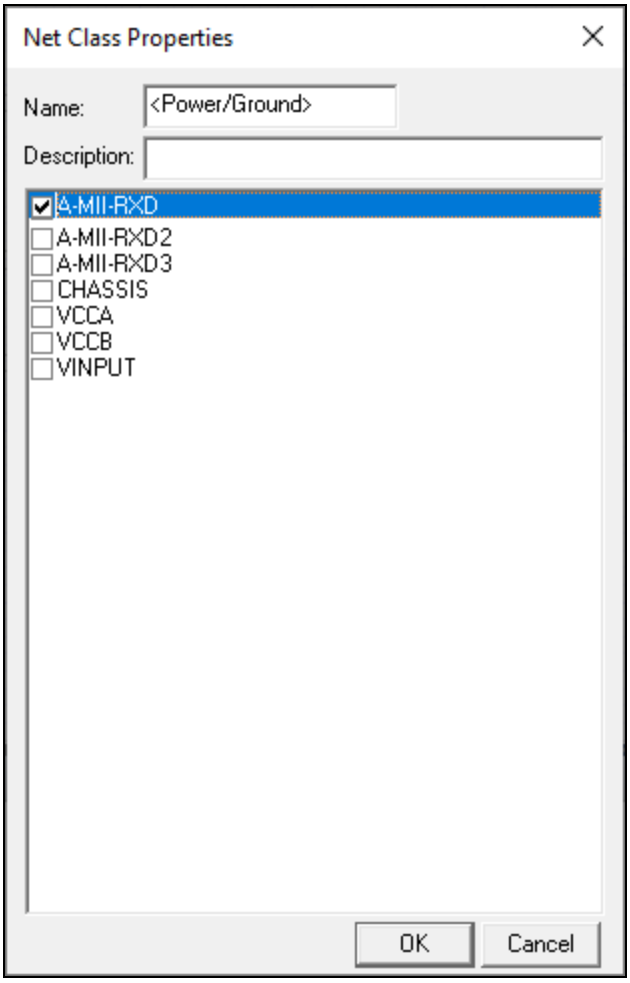
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Net Class Properties

Selecting **Edit** from the right-click menu opens the **Net Class Properties** window. Use this window for net class modification.

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



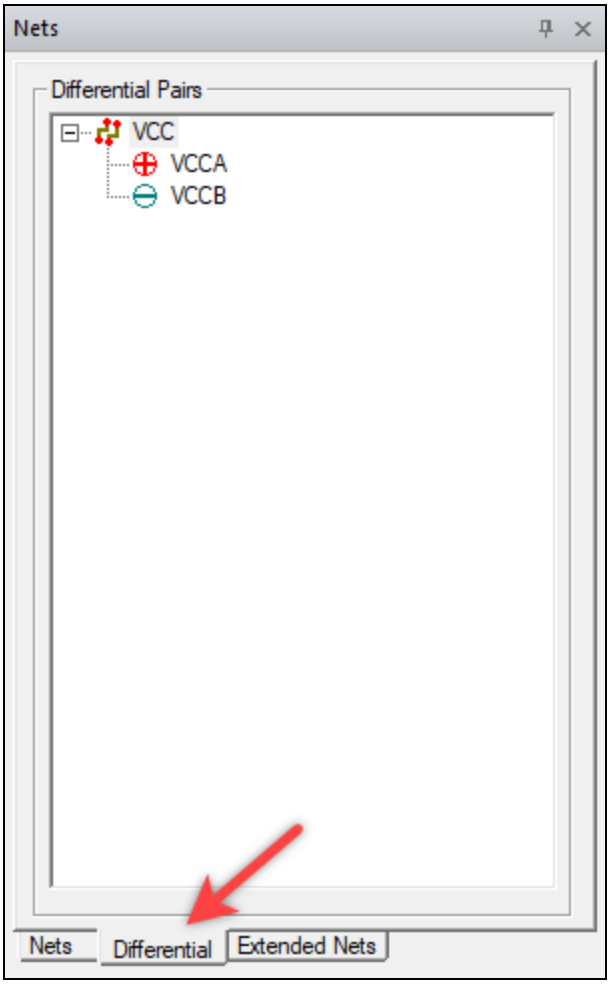
Differential Pairs

Each Differential Pair is composed of a positive net and a negative net. From the **Nets** window, you can view and create Differential Pairs.

Viewing Differential Pairs

Click the **Differential** tab to display a list of Differential Pairs.

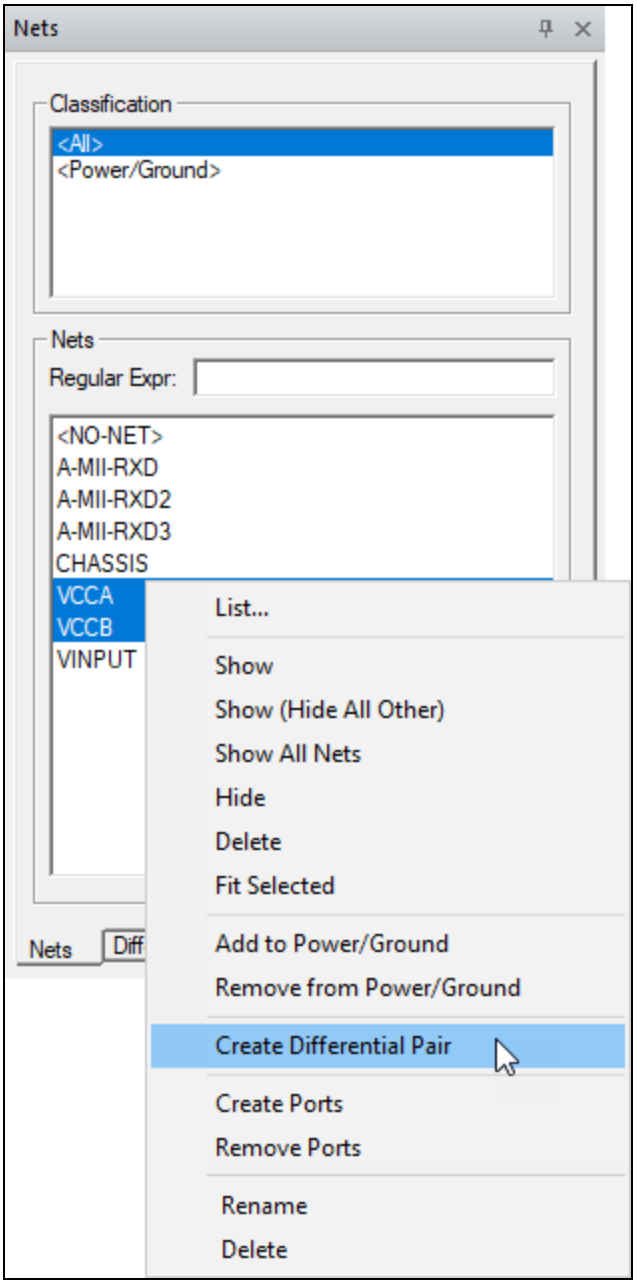
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating Differential Pairs

Select two nets in the **Nets** list and right-click to create a new Differential Pair.

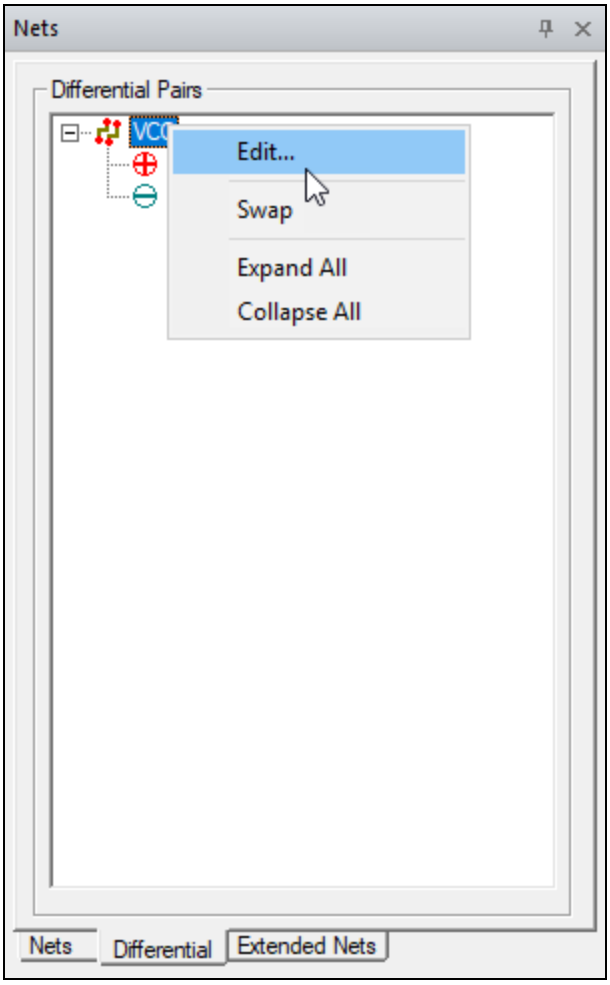
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Differential Pairs** list, you can select a pair, open the right-click menu and select:

- **Swap** – swaps the positive and negative net.
- **Edit** – opens the **Edit Differential Pairs** window.
- **Expand All** – expands all Differential Pairs.
- **Collapse All** – collapses all Differential Pairs.

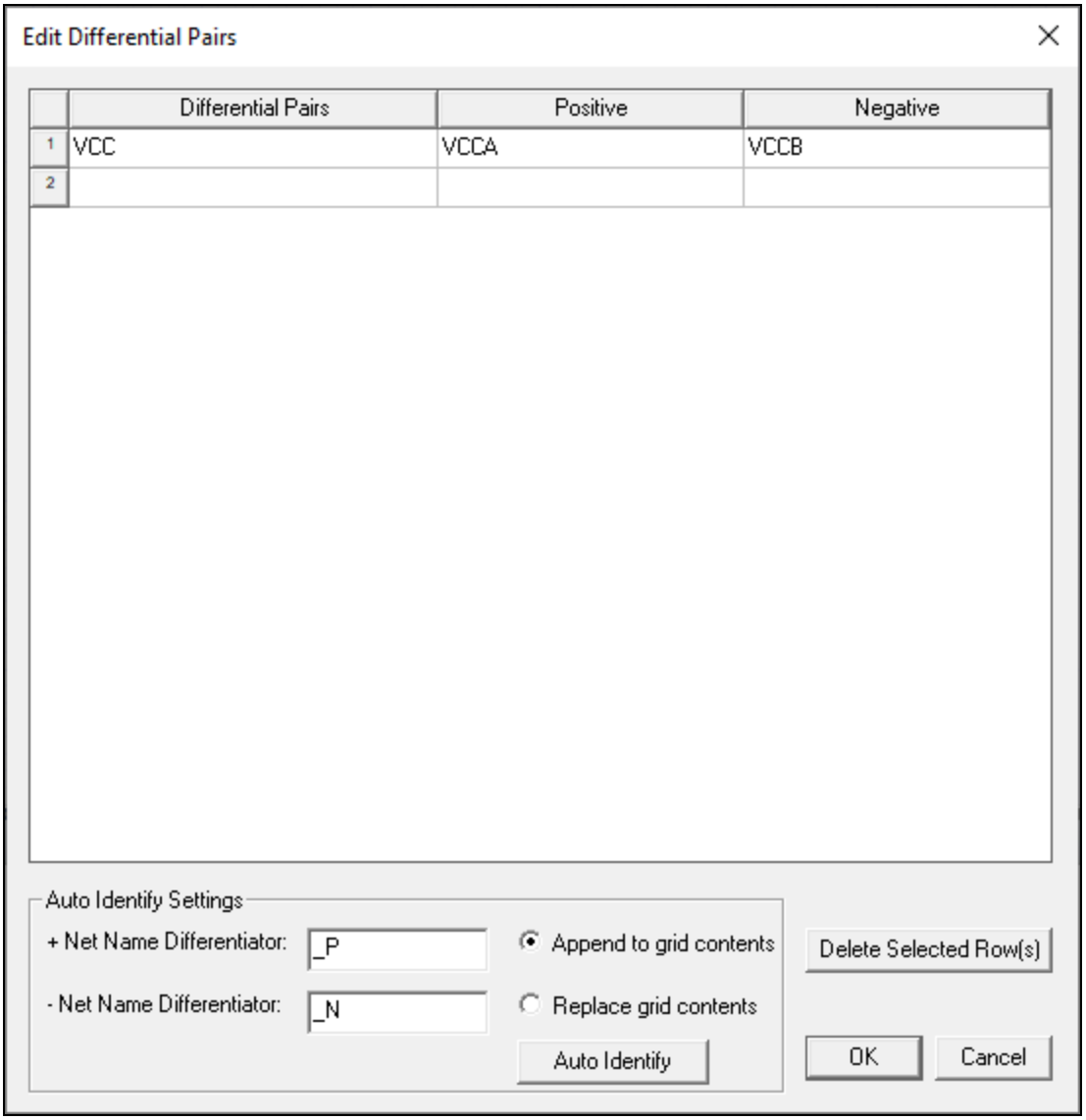
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Edit Differential Pairs Window

From the **Edit Differential Pairs** window, you can delete pairs or auto detect Differential Pair with postfixes of the positive and negative nets that are provided. Enter differentiators to identify the positive and negative parts of pairs. Use **Append to grid contents** to add any new pairs to the list in the grid and **Replace grid contents** to replace existing pair with newly identified pairs.

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



Extended Nets Tab

Extended nets are used to indicate a logical equivalence among multiple nets. This is useful when two nets can be considered electrically equivalent, but are not physically connected. For example, routing for a data signal may have a short trace between a series termination resistor and pin. While the net assignment for traces on either side of the resistor are different, logically they are equal and can therefore be grouped into an extended net.

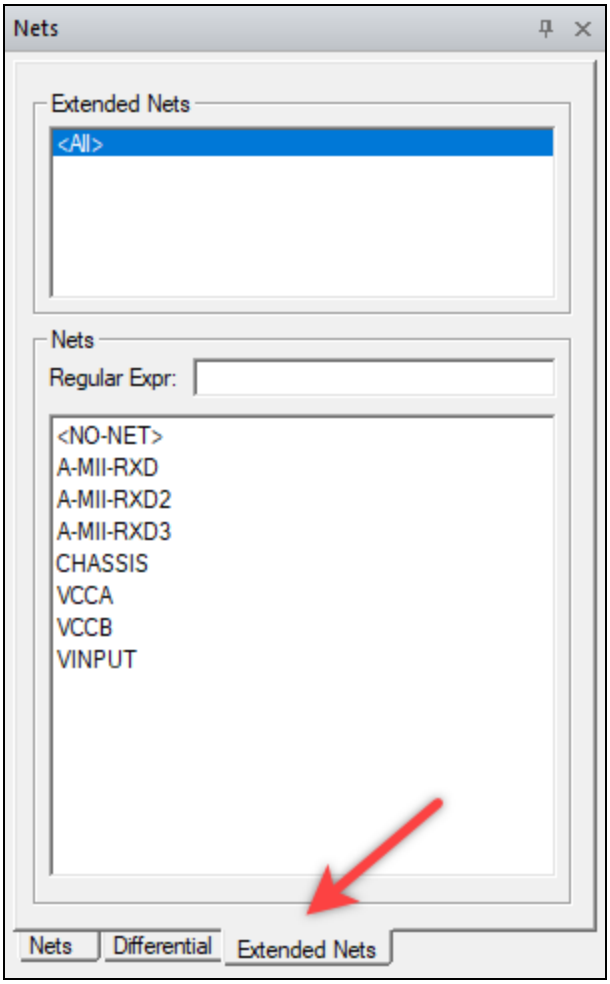
Extended nets are collections of nets that are similar to net classifications but with subtle differences. A net can be in only one extended net at a time, in contrast to being in multiple

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

classifications simultaneously. There are also no default extended nets and no ports are associated with extended nets.

Extended nets are used for SIwave SYZ with HFSS Regions simulations. If an HFSS region contains capacitors, inductors, or resistors that connect two different signal nets, these signal nets must be defined as extended nets. SIwave needs this definition to know how to pull in required nets and auto generate ports in the region.

The **Extended Nets** tab contains two panes: an upper **Extended Nets** pane and a lower **Nets** pane.



Extended Nets Pane

The **Extended Nets** pane lists all extended nets. Use it manage the extended nets. When an extended net is selected in the **Extended Nets** pane, the nets assigned to that extended net appear in the **Nets** pane. The entry <All> lists all nets in the **Nets** pane including entities not in a

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

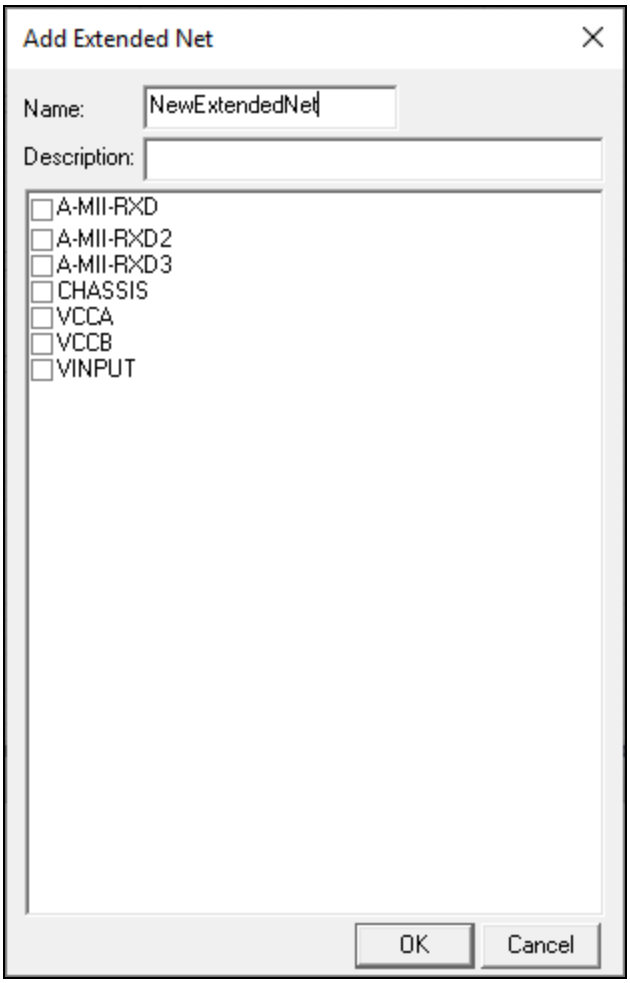
net. <All> is not an extended net. The **Extended Nets** pane's shortcut menu offers the following actions:

- **New** – creates a new extended net.
- **Auto Identify** – automatically identifies new extended nets in the **Auto Identify Settings** window.
- **Edit** – changes the selected extended net.
- **Delete** – deletes the selected extended nets.
- **Select** – selects the members of the extended nets in the Layout editor.
- **Show** – shows the members of the selected extended nets in the Layout editor if hidden.
- **Show (Hide All Other)** – shows the members of the selected extended nets in the Layout editor while hiding all other nets.
- **Hide** – hides the members of the selected extended nets in the Layout editor.

Create an Extended Net

1. In the **Extended Nets** pane, right click and click **New** to open the **Add Extended Net** window.

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

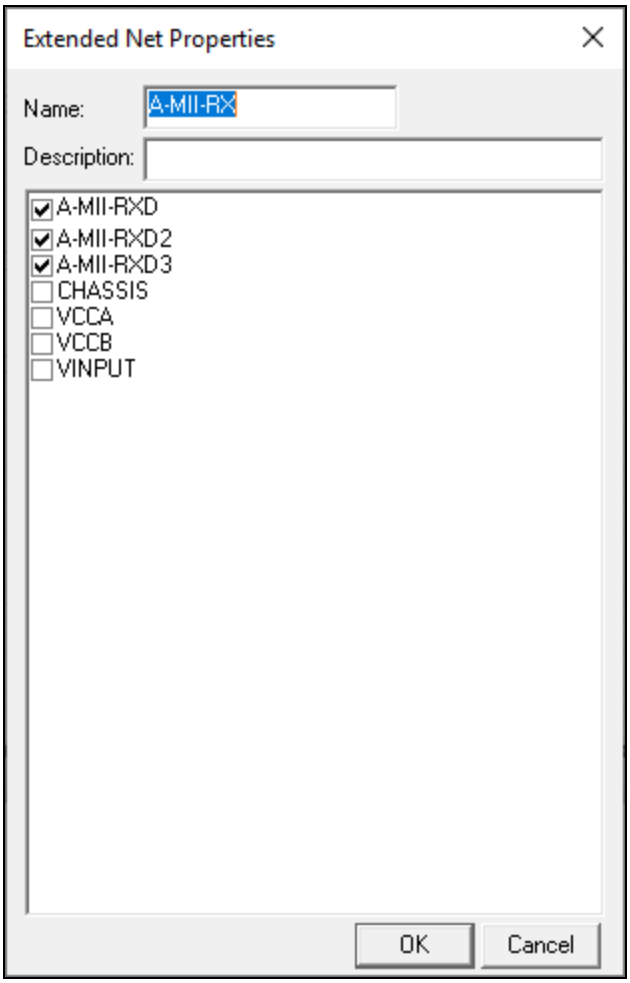


2. Enter a name and optional description.
3. Select the nets that will make up the extended net.
4. Click **OK**.

Editing an Extended Net

1. In the **Extended Nets** pane, right click and select **Edit** to open the **Extended Net 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.

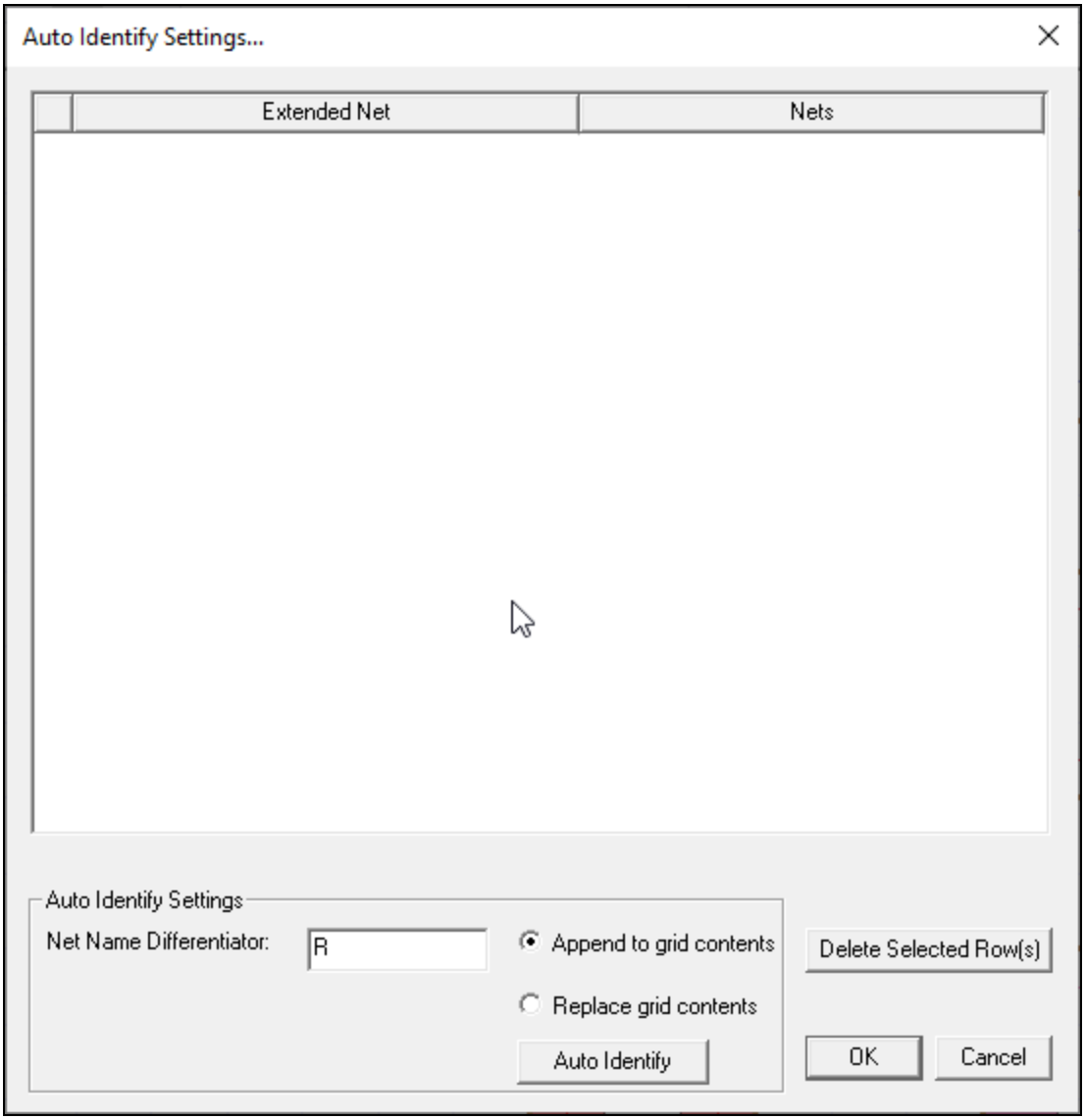


2. Change the name and optional description.
3. Select the nets that will make up the extended net.
4. Clear the nets that should not be in the extended net.
5. Click **OK**.

Auto Identify Settings window

From the **Auto Identify Settings** window, you can automatically detect and create extended nets, rename extended nets, or delete extended nets. Existing nets are listed in the grid. Enter a differentiator to identify extended nets. Use **Append to grid contents** to add any new extended nets to the list in the grid and **Replace grid contents** to replace existing extended nets with newly identified ones.

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



Nets Pane

The **Nets** pane displays the members of the extended nets that are selected in the **Extended Nets** pane. These nets can be filtered using regular expressions. In the **Nets** pane, the short menu offers these actions:

- **Show** – shows the selected nets in the Layout editor if any are hidden.
- **Show (Hide All Other)** – shows the selected nets in the Layout editor while hiding all other nets.
- **Show all nets** – shows all nets in the Layout editor.
- **Hide** – hides the selected nets in the Layout editor.

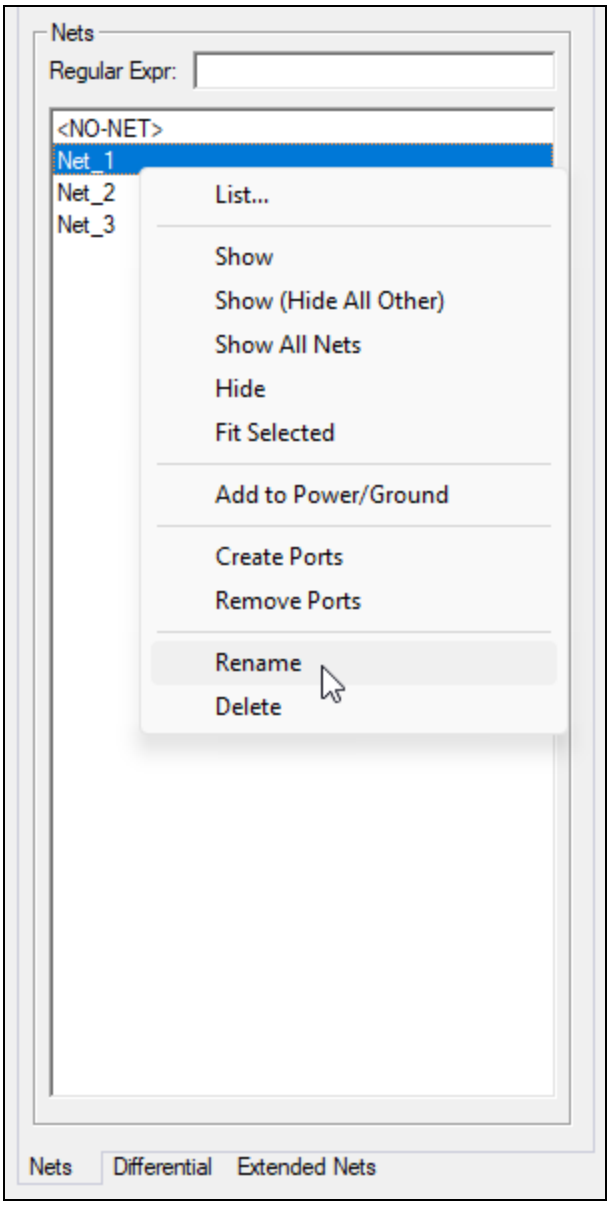
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Delete** – deletes the selected nets from the project. Members of the net remain in the project.
- **Fit Selected** – zooms the Layout editor into show the selected nets.

Rename

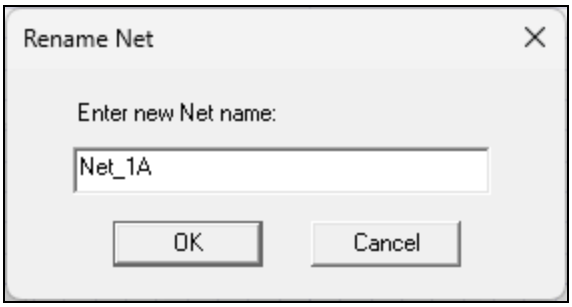
The **Rename** command allows you to change the name of a selected net.

1. From the **Nets** pane, right-click on a net and select **Rename** to open the **Rename Net** window.



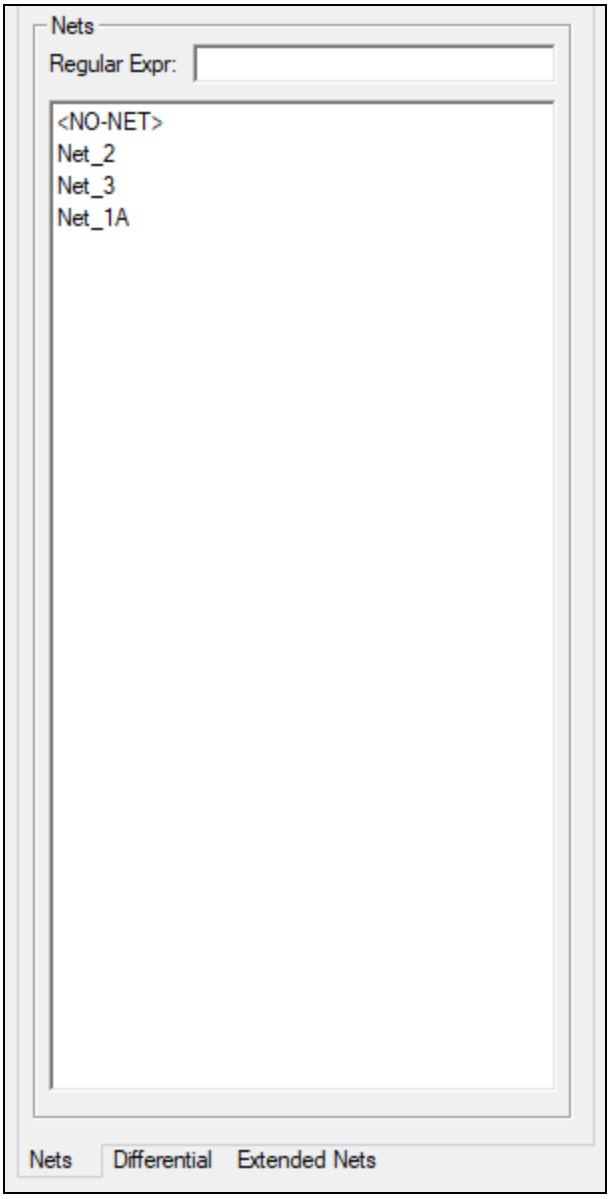
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter a **new Net name** in 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.

3. Click **OK** and the new name appears in the list.



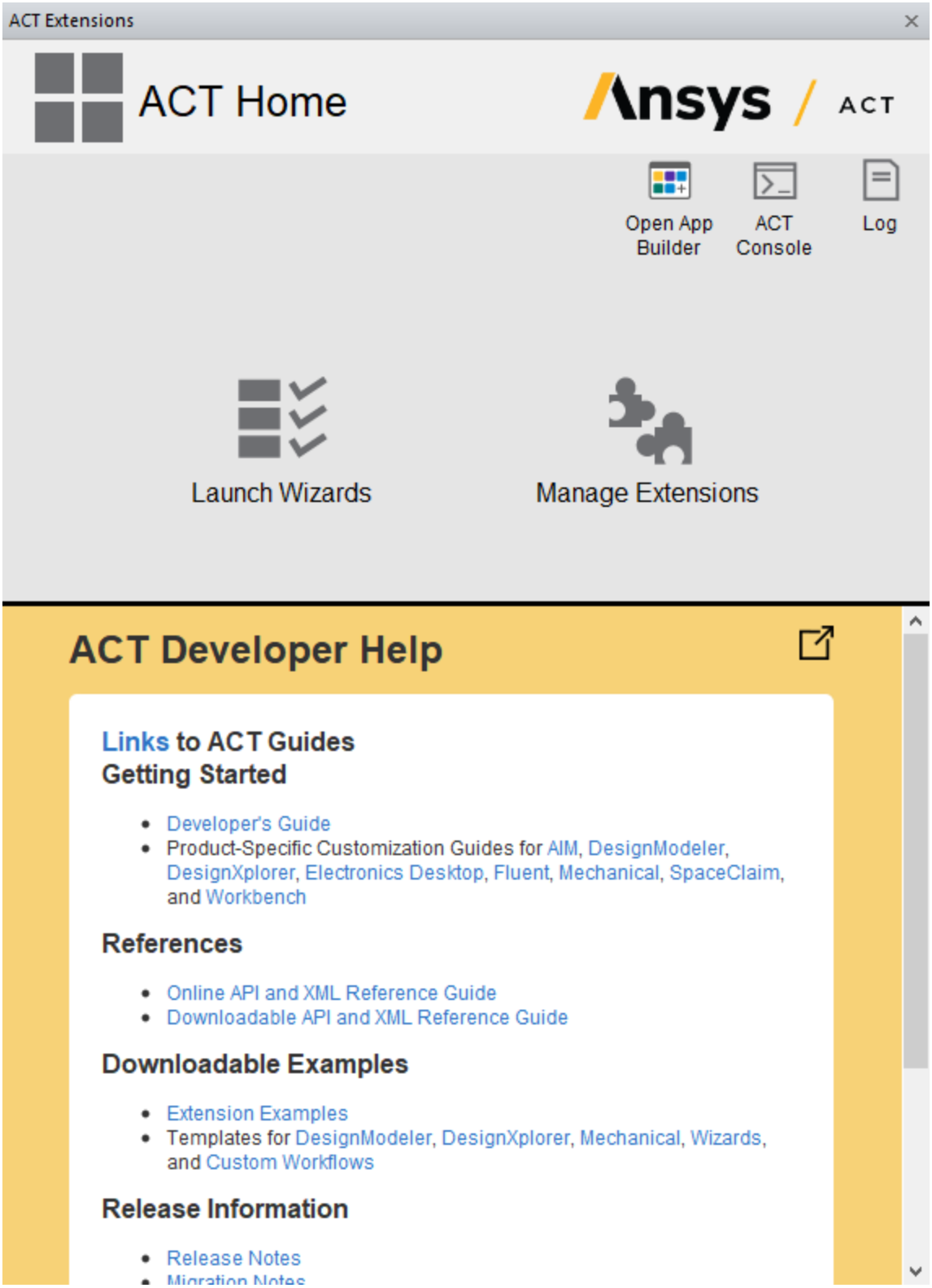
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

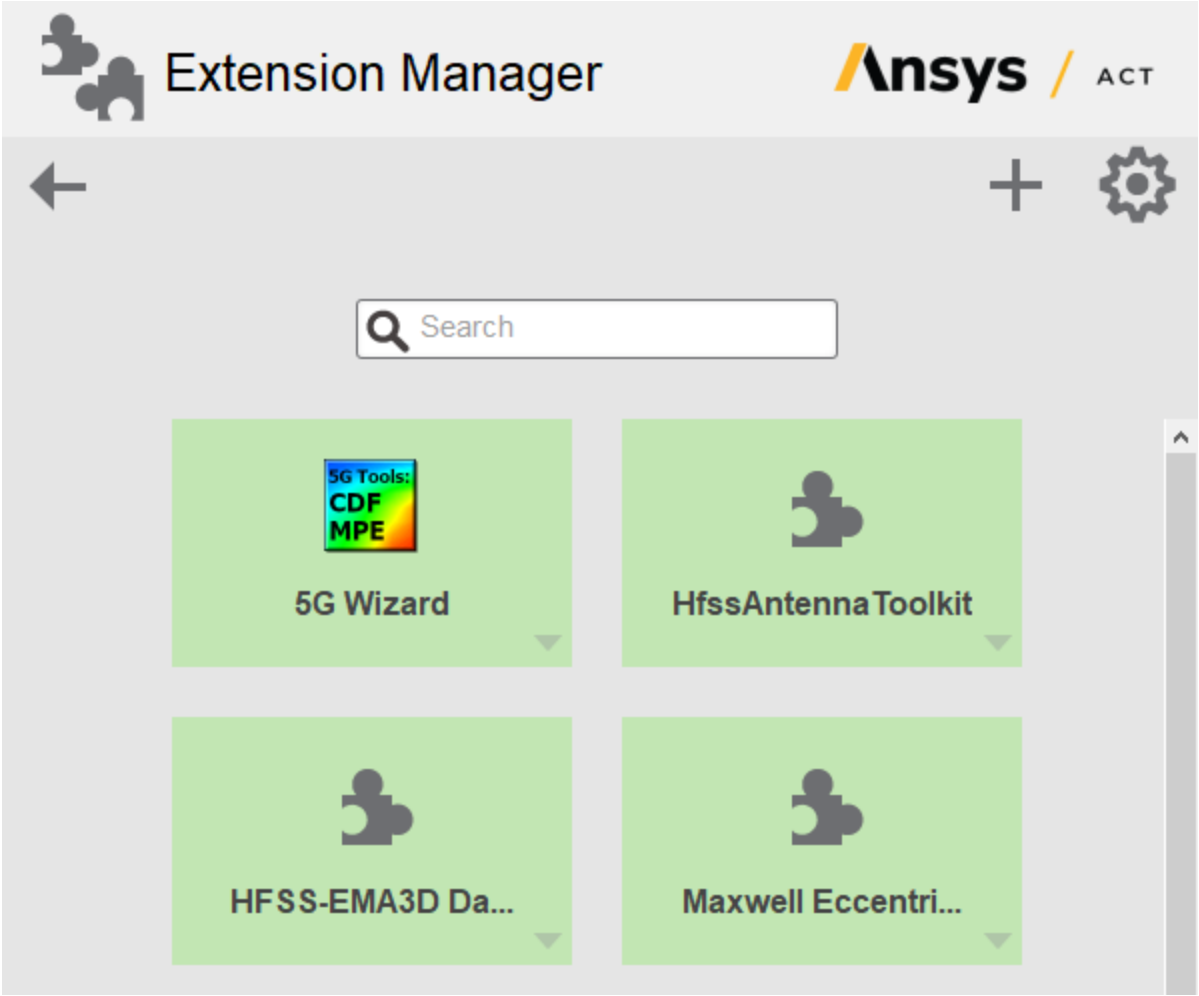
The **View** > **ACT Extensions** command opens the ACT Extensions window for toolkit extensions implemented via Ansys ACT. Depending on how you size and arrange the desktop, the appearance may vary. You may choose to detach and size the window to suit your needs.

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



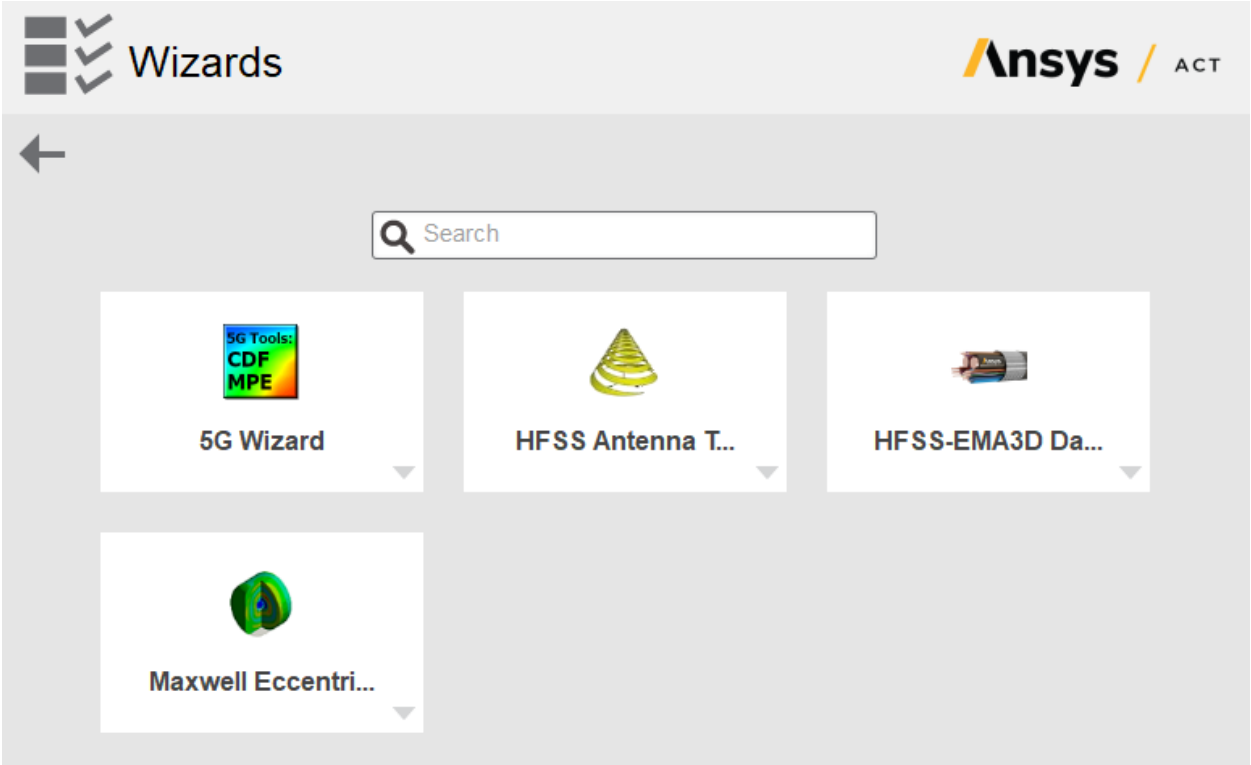
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



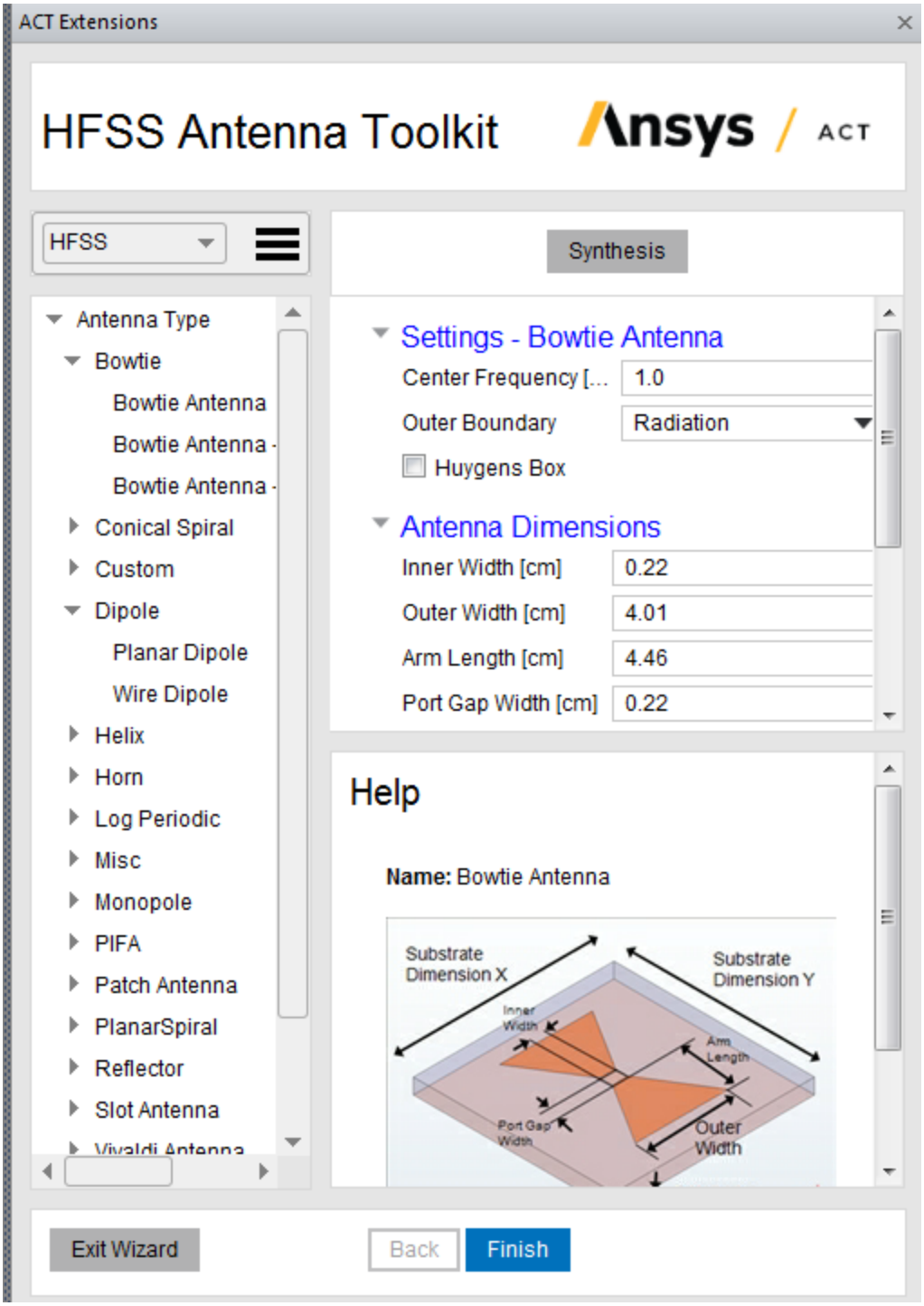
Click the **Launch Wizards** icon to load existing toolkits for activation. If there are many available extensions, you can use the **Search** bar. Search is not case sensitive.

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.

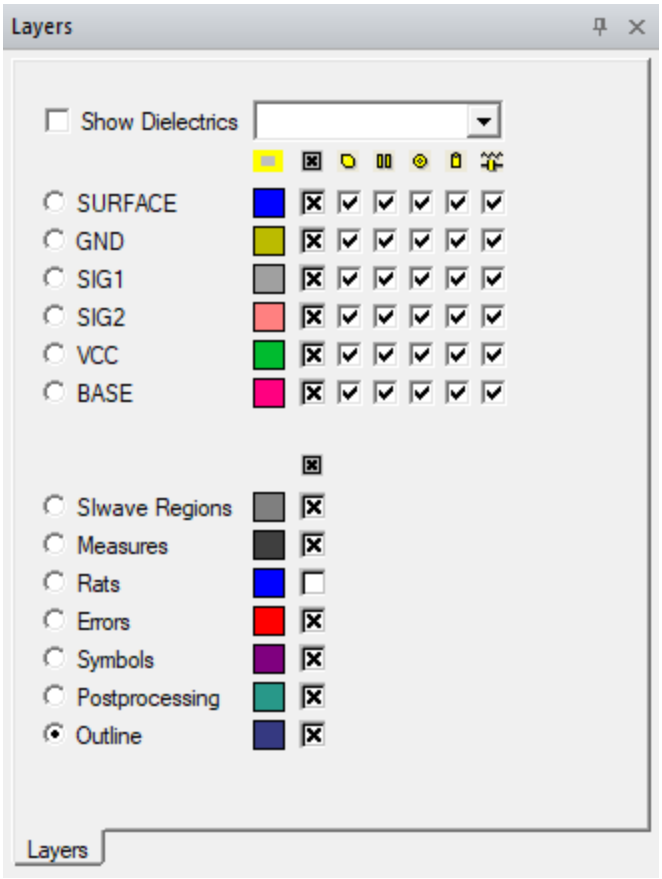
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

Working with the Layers Window

In the **Layers** window, you can select the working layer and the visibility of layout objects on specific layers, objects such as shapes, pads, components, nets, and the layers themselves. The **Layers** window is divided into three sections: stackup layers, non-stackup layers, and a preprocess mesh section. This dockable window can be resized and relocated.



- To show or hide the **Layers** window on the desktop, click **View** > **Layers**. A check box appears next to this command if the **Layers** window is visible.
- To select a working layer, click the radio button of the layer you want to work on.
- To view specific items on different layers, use **Show Dielectrics**, **Zone Filter**, the layers check boxes, and View Name drop-down menu.

If the stackup is a multizone stackup, select a zone from the **Zone Filter** to see the available layers.

The **Show Dielectrics** check box controls whether dielectric stackup layers are shown. If unchecked, dielectric layers are hidden and holes (vias and pins) are rendered continuously visible from the top-most visible signal layer to the bottom most visible layer. If checked,

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

dielectric layers appear and hole visibility can be finely controlled by each layer (but not necessarily continuously visible).

The View Name drop-down menu lists user-defined views and their controls. You can configure how layers and nets are displayed and then save this visibility as a view. Later, select that view from the drop-down menu to reset the **Layout** window to it. The view controls and default options are:

- **<Surface>** shows the top and bottom conducting layers of the layer stackup and hides all other layers. This is the default view.
- **Save Current View** opens a dialog that allows you to save the current setting of layers and nets to a new layout view.
- **Delete View** opens a dialog that allows you to delete saved layout views.

Check boxes control what is displayed in a layer. Icons at the top of a column control the entire column. Individual check boxes control the item visibility for the layer corresponding to the check box's row. Hover over an icon to see what the column controls:

Name	Icon	Controls the visibility of	Appears
Fill/Unfill All		Color in the object	Always
Show/Hide All		All objects	Always
Shapes		Shapes	Always
Lines		Lines and paths	Always
Pads		Pads	Always
Holes		Holes	Always
Components		Components	Always
Mesh		Mesh of signal layers	When a preprocess mesh is displayed
Background Dielectric Mesh		Mesh of dielectric layers	When a preprocess mesh is displayed with a laminate stackup

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

A preprocess mesh section may appear below the non-stackup layer section. It controls design-wide visibility of:

- Airbox non-mesh and mesh
- Hole, solderball, and bondwire mesh

The visibility of these objects is not restricted to their layers. Also, layer visibility check boxes cannot control these mesh visibilities. Non-mesh visibilities are controlled by the layer check boxes.

Working with the Measure Window

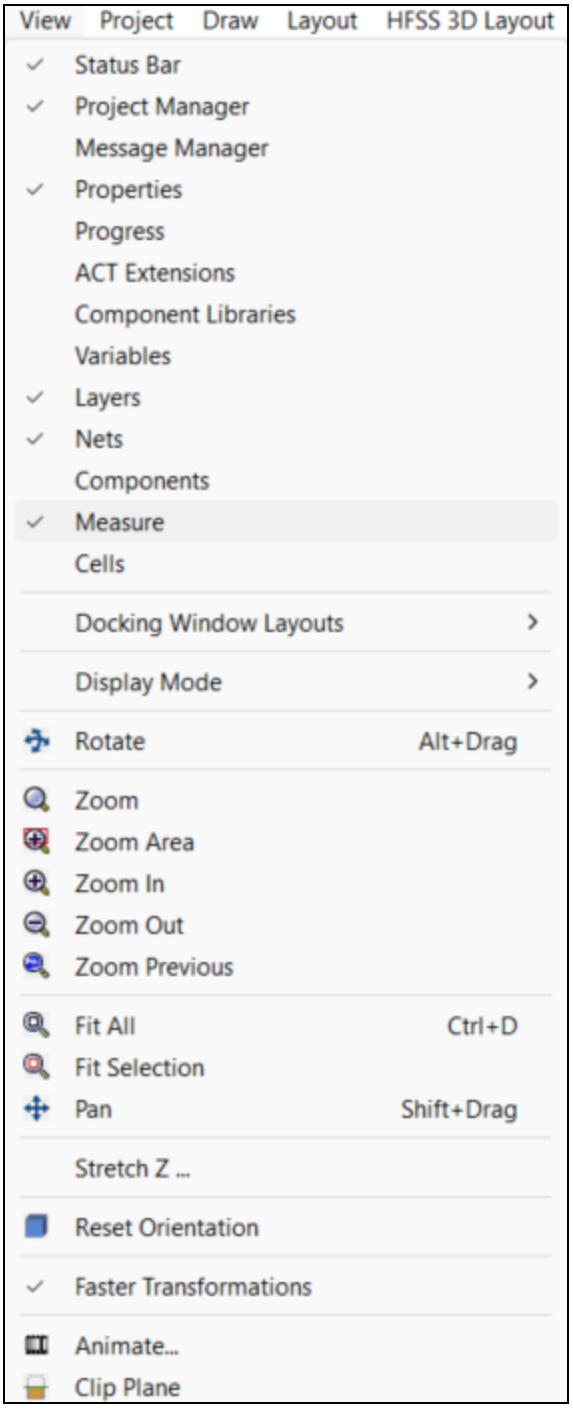
Use the **Measure** window to measure the position, length, and area of elements in an HFSS 3D Layout design.

> **Note:**
>
> Refer to the steps below to use the Measure window toolset. Users will not be able to gather measurement data from encrypted components. Certain parameters of encrypted technologies will also be obfuscated.

Accessing Measure Mode and Viewing Measured Data

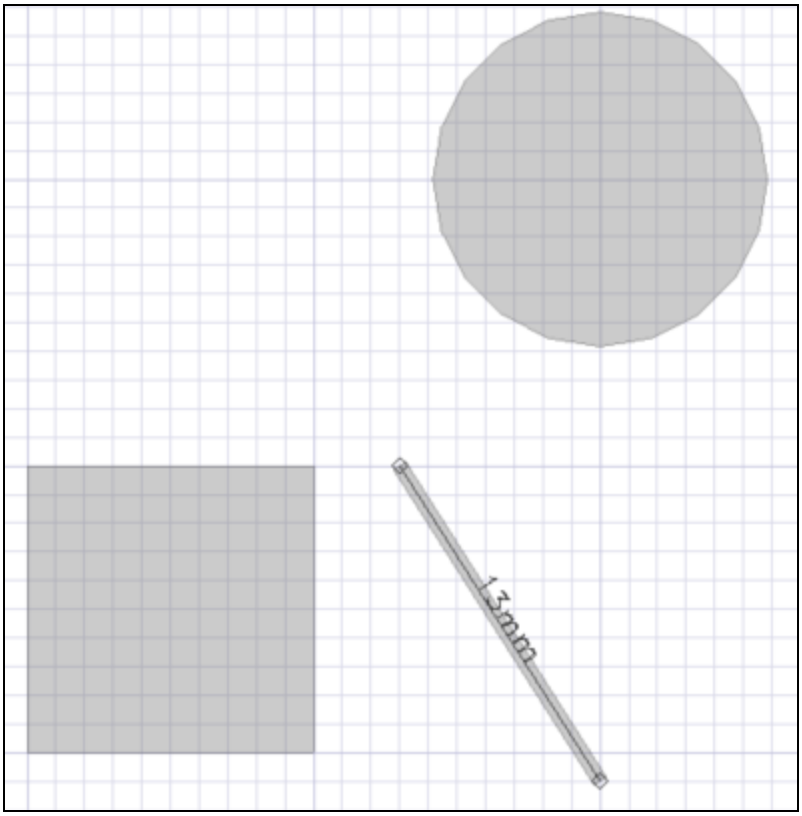
To show or hide the **Measure** window, click **View** > **Measure**. A check box appears next to **Measure** if the **Measure** window is visible.

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



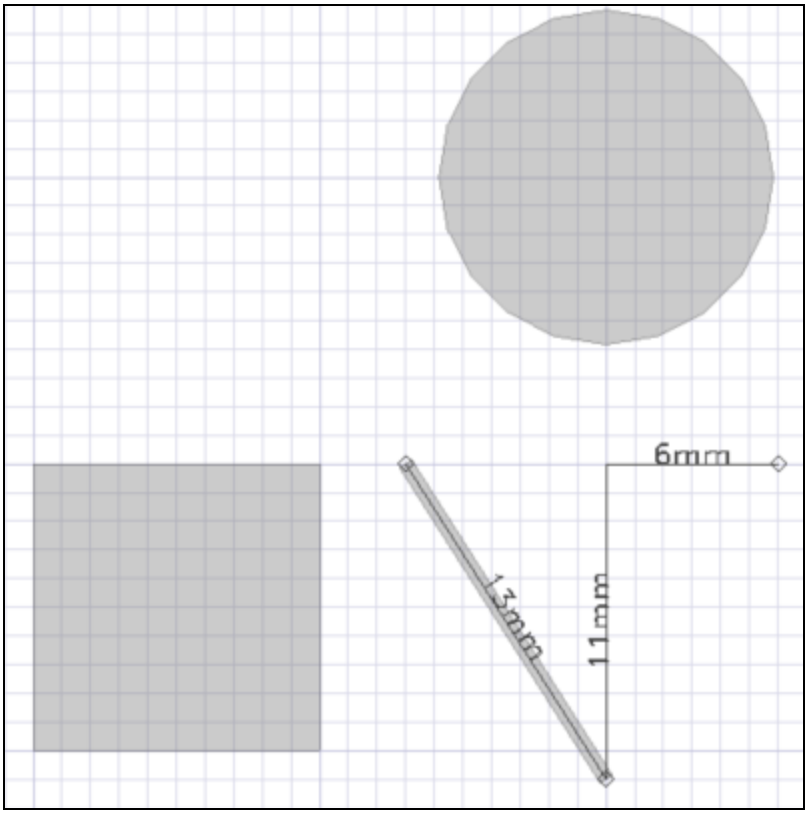
Measure mode allows users to select any two grid points in the **Layout Editor** to define as a segment and display the linear distance between them (i.e., the grid points become end points). The subsequent measurements are displayed beside the segment.

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



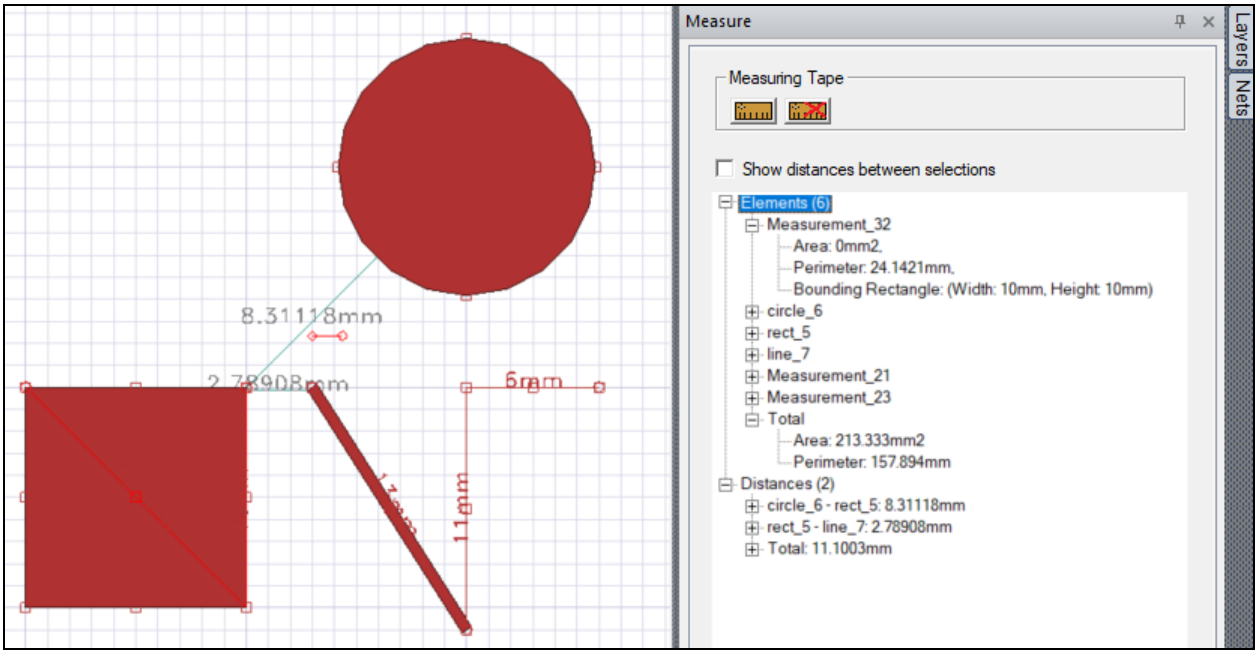
Users can add more grid points to define multiple segments. The distance between each point is measured separately. The individual measurements are displayed beside each segment in the **Layout Editor**.

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

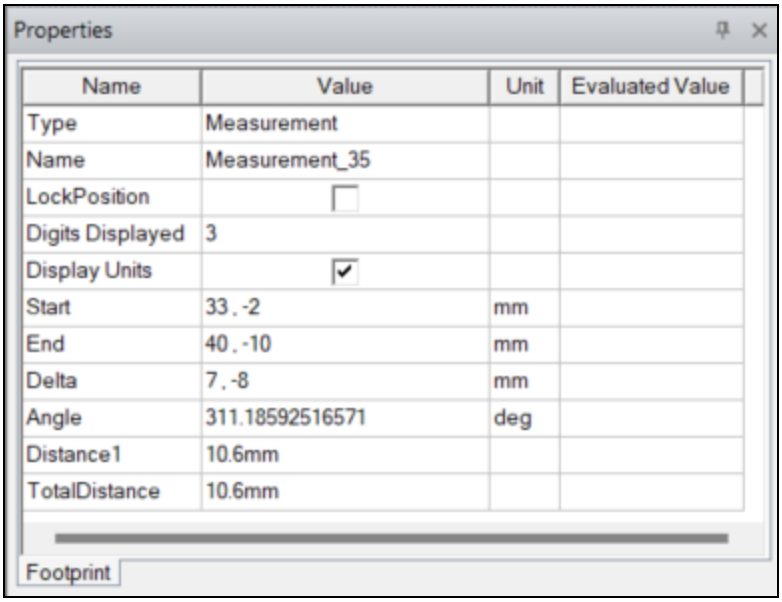


Highlighted (i.e., selected) measurements are also displayed in a parameter tree in the **Measure** window, separated by **Element(s)** and measured **Distance(s)**, and then by individual measurement. Each measurement is described by a combination of area, perimeter, and length, and any bounding device by width and height.

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



Selecting measurements also populates the **Properties** window with the following parameters:



- **Type** — designates the types of values shown (i.e., **Measurement**(s))
- **Name** — identifies the specific measured element or segment (e.g., **Measurement_35**)
- **LockPosition** — check the box to activate Position Locking (unchecked 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.

- **Digits Displayed** — enter a value to set the number of significant digits displayed (default is **3** significant digits)

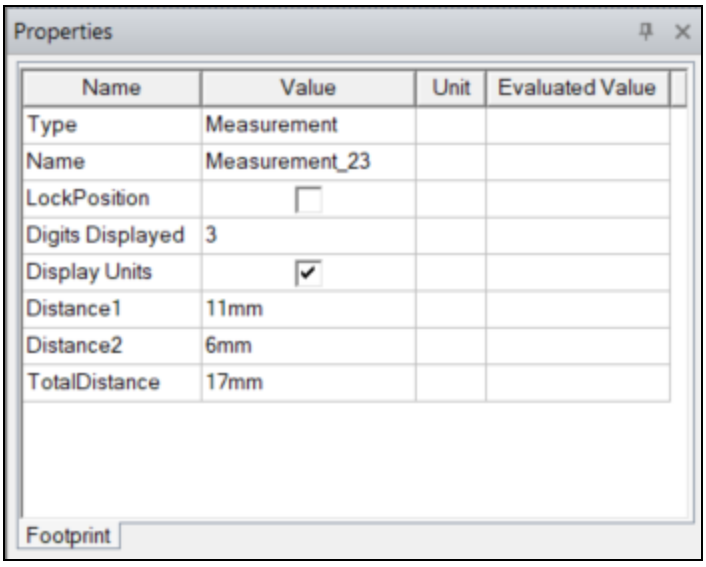
> **Note:**
>
> **Measurement Distance Displays** can be configured from the **Layout Options** window > **Object** tab.

- **Display Units** — check the box to display measurement distances in the **Layout Editor** (unchecked by default)

> **Note:**
>
> Default display units can be configured from can be configured from the **General Options** window > **Default Units** tab.

- **Start**, **End**, **Delta**, and **Angle** — editable position values which display/alter the measured segment in the **Layout Editor**
- **TotalDistance** — displays the length of the segment (e.g., **10.6 mm**)

Multi-segment measurements are displayed in the **Properties** window as separate uneditable parameters (e.g., **Distance1**, **Distance2**, **Distance3**, et cetera).



Using Measure mode

Complete these steps to use **Measure** mode.

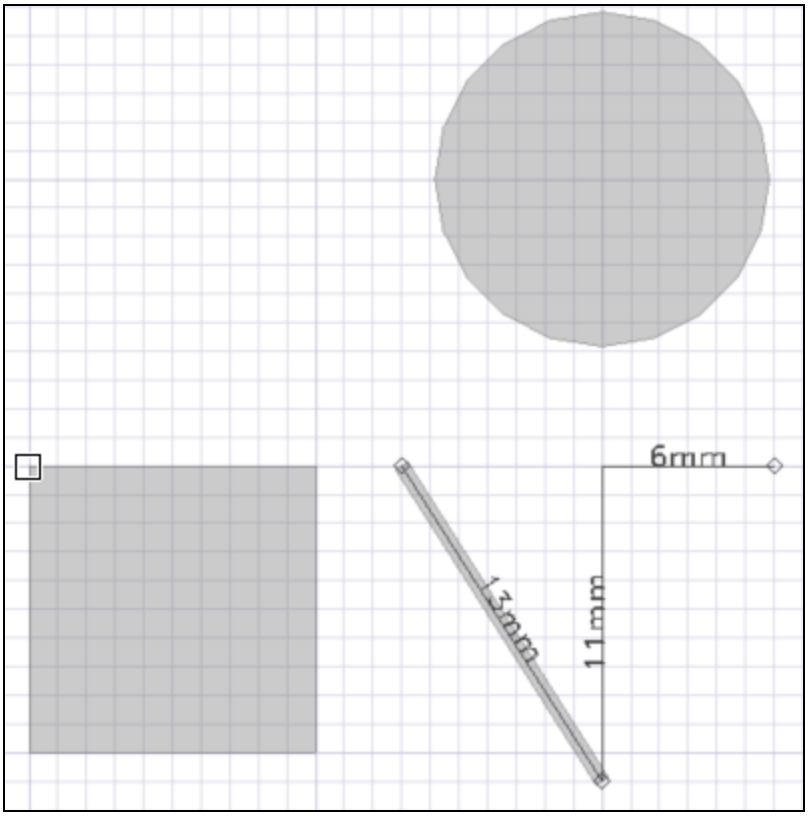
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Options displayed in menus differ depending on whether **Electronics Desktop** is configured for **General Mode** or **IC Mode**. Refer to **Switching to IC Mode**.

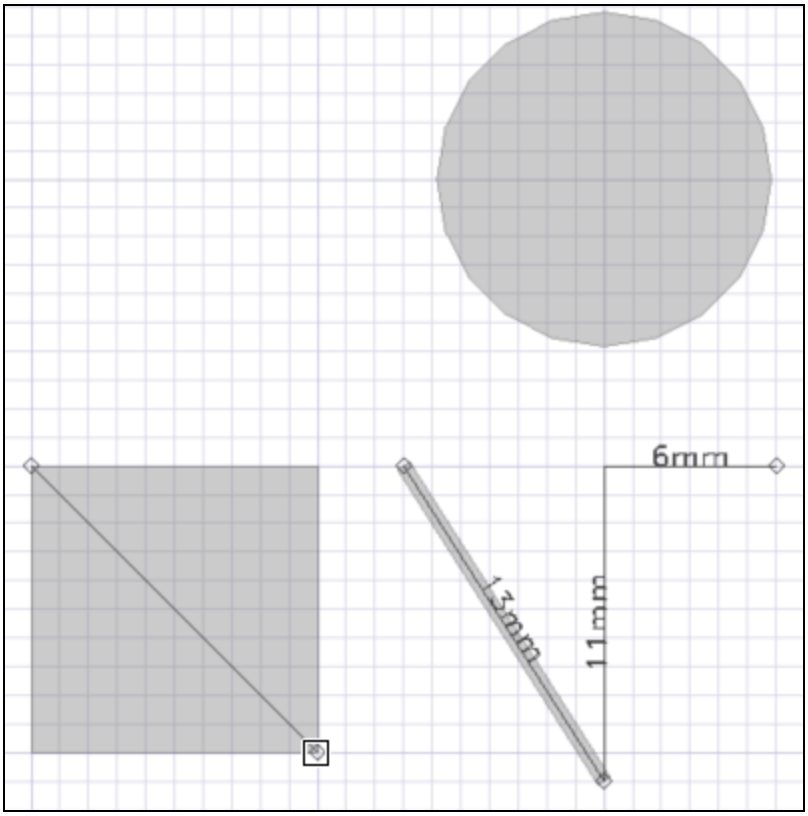
1. From the **Measuring Tape** area, click the **Measuring Tape** icon.

The cursor automatically changes shape in the **Layout Editor** when **Measure** mode is active.



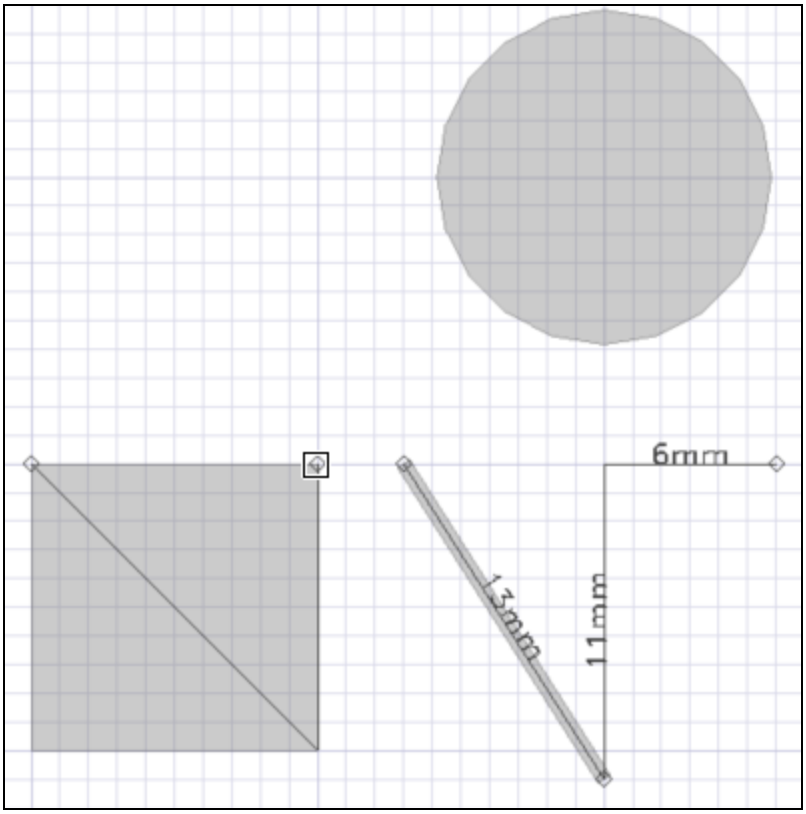
2. Click twice to select two points in the **Layout Editor** to define a segment.

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 more points, as appropriate.

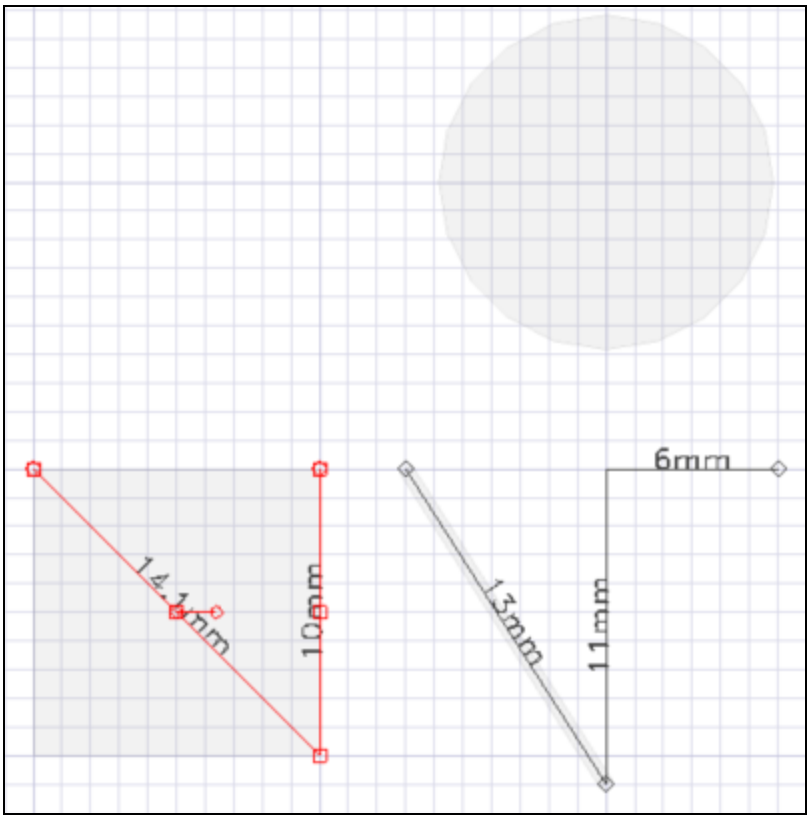
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. To complete the measurement and measure new segments without exiting **Measure** mode, do one of the following:

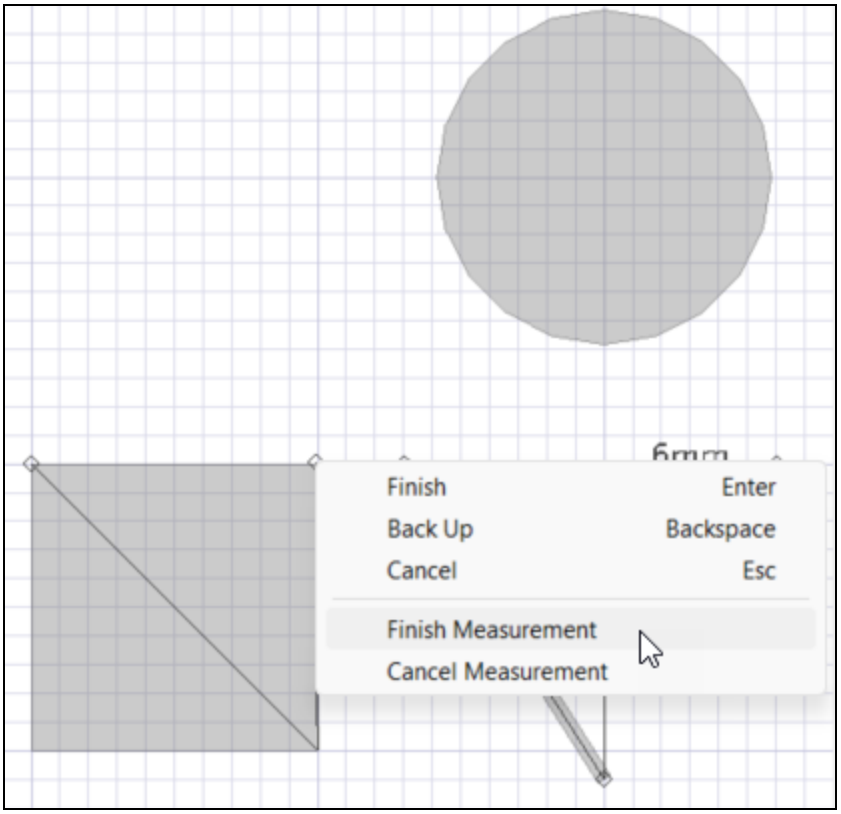
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Double-click the last point to complete the current measurement.



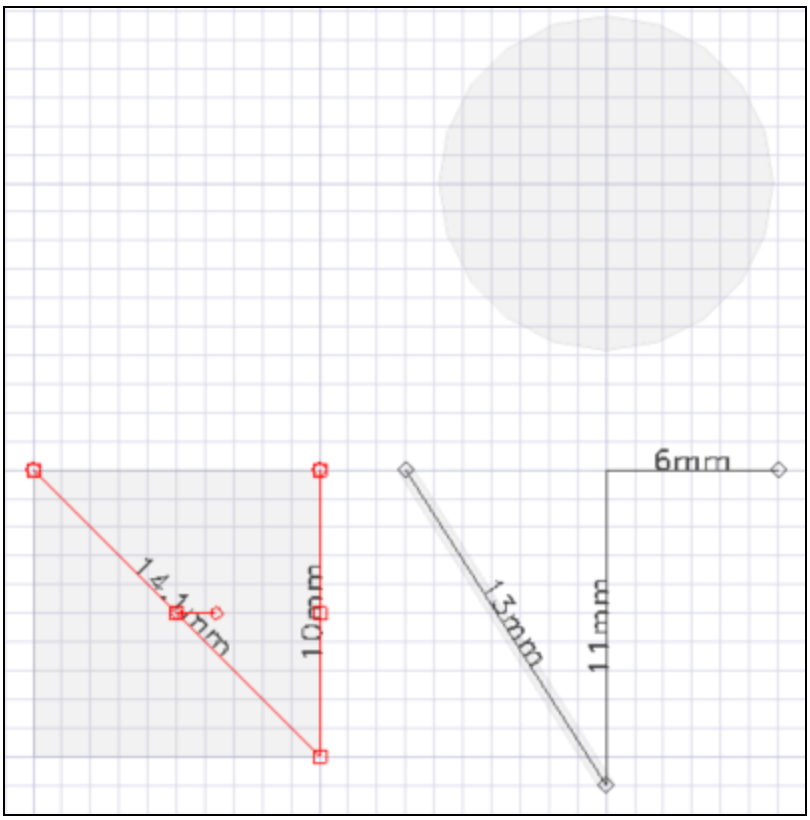
- Right-click within the **Layout Editor** and select **Finish Measurement**.

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



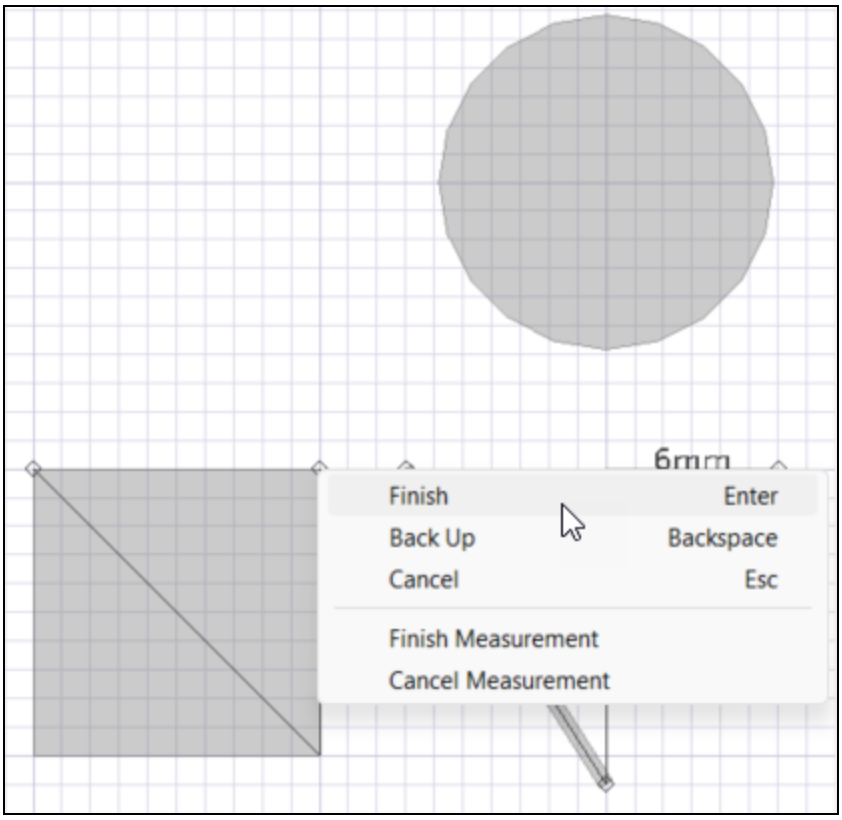
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To complete the measurement and exit **Measure** mode, do one of the following:
    - Click **Enter** to complete the segment(s) and view their measurements.



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 within the **Layout Editor** and select **Finish**.



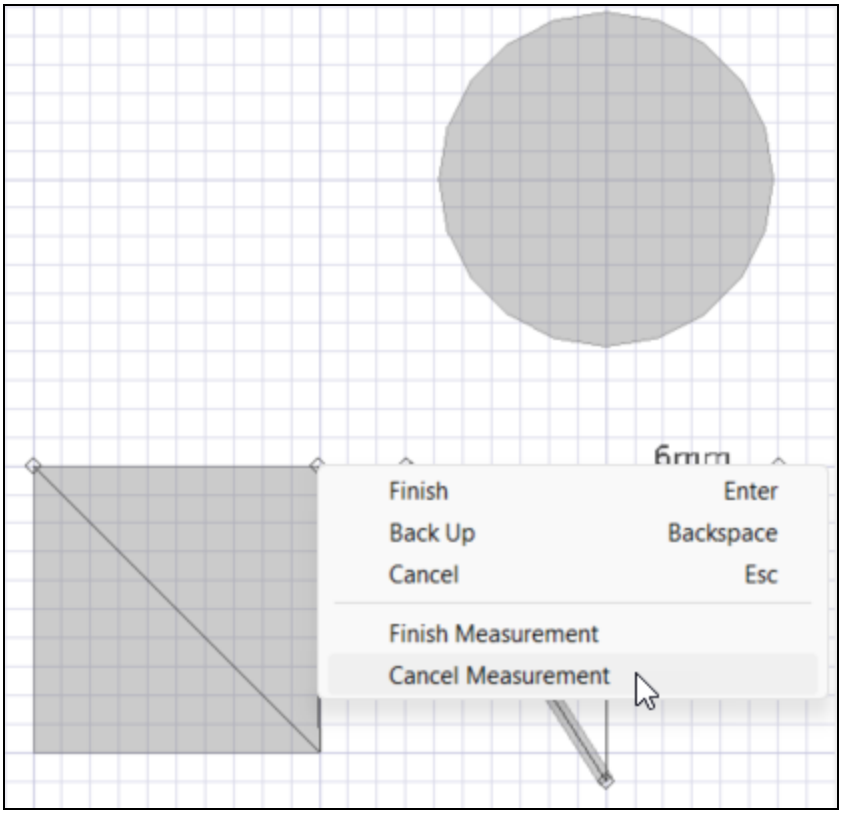
The new segment (or multi-segment) length(s) display in the **Layout Editor** and the **Properties** window.

> **Note:**
>
> While segment measurements are displayed in the **Layout Editor**, they can be moved, rotated, or flipped. Measurement handles may be edited, but measurements can not be copied or pasted.

Canceling Measurements

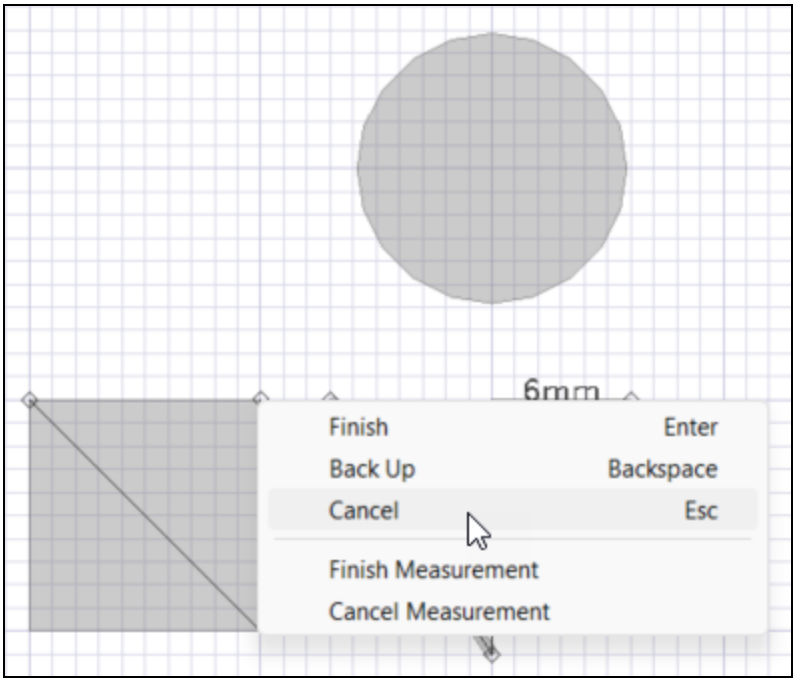
To cancel a measurement without exiting **Measure** mode, right-click in the **Layout Editor** and select **Cancel Measurement**.

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



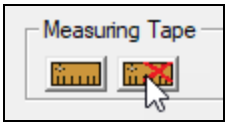
To cancel a measurement and exit **Measure** mode, right-click in the **Layout Editor** and select **Cancel**.

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



Erasing Measurements

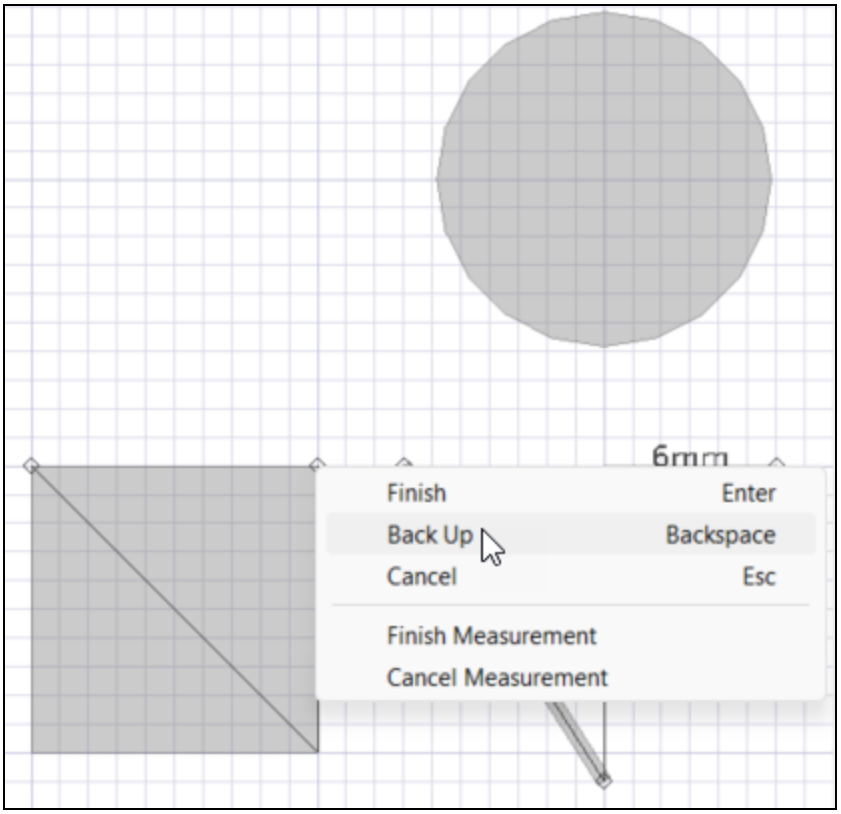
To clear the measurements from the **Layout Editor**, click the **Erase Measurements** icon.



Backing Up Measurements

To back up the current measurement, right-click in the **Layout Editor** and select **Back Up**.

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

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

	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.

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 (⇒)

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

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:

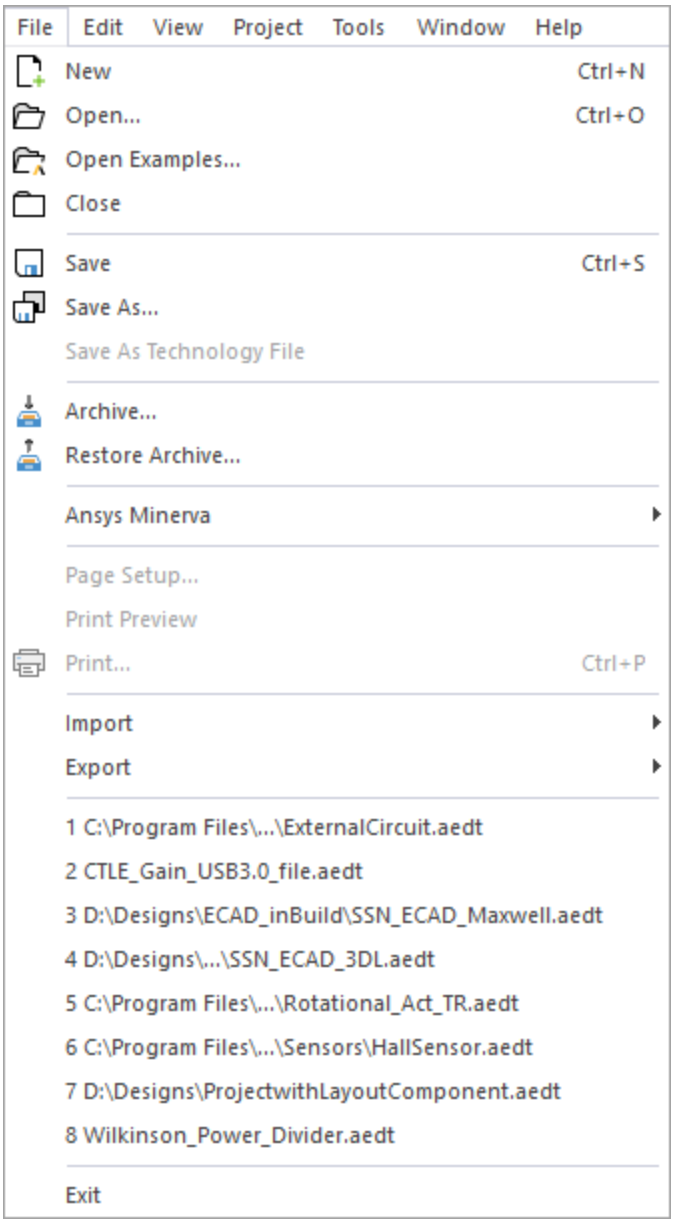
File Edit View Project Draw [Modeler | Layout | Schematic](1) <*Design-Type*>(2) Tools Window Help

> **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.

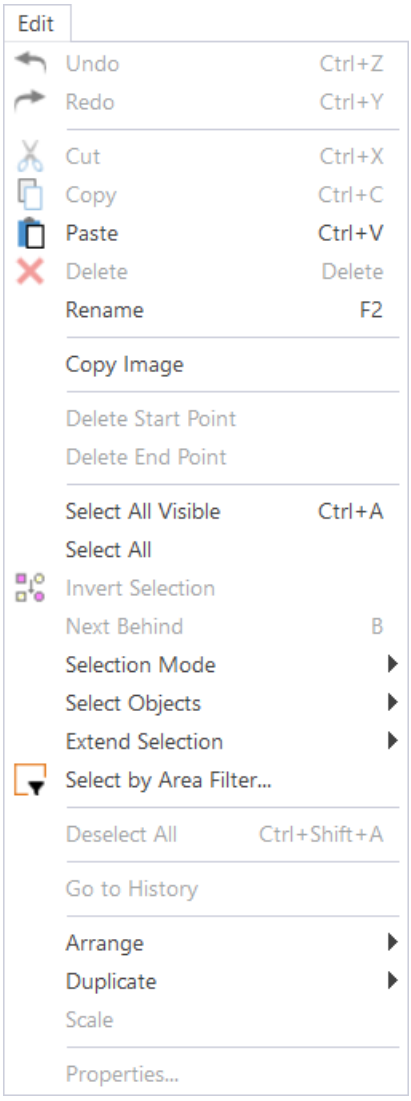
These menus include:

- 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.

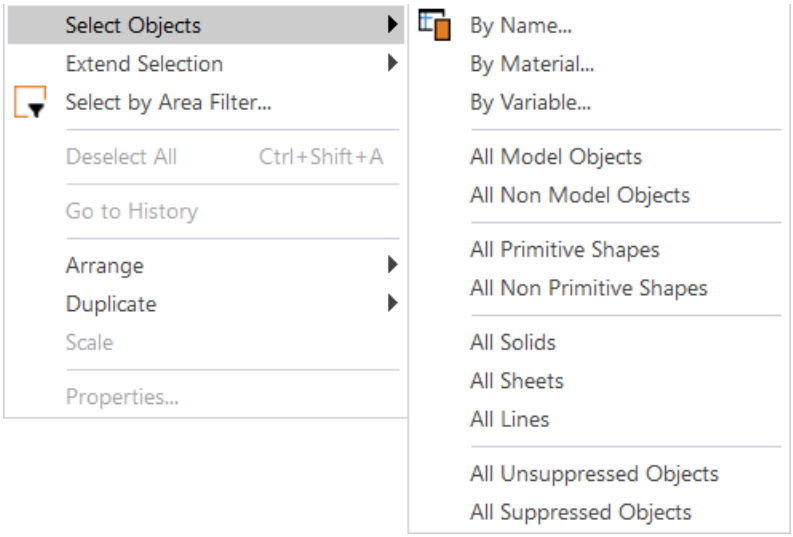


Commands will be available or not available for clicking depending on what is currently selected or on whether any operations have been performed, undone, or redone. The 3D Modeler's Edit menu contains the following options:

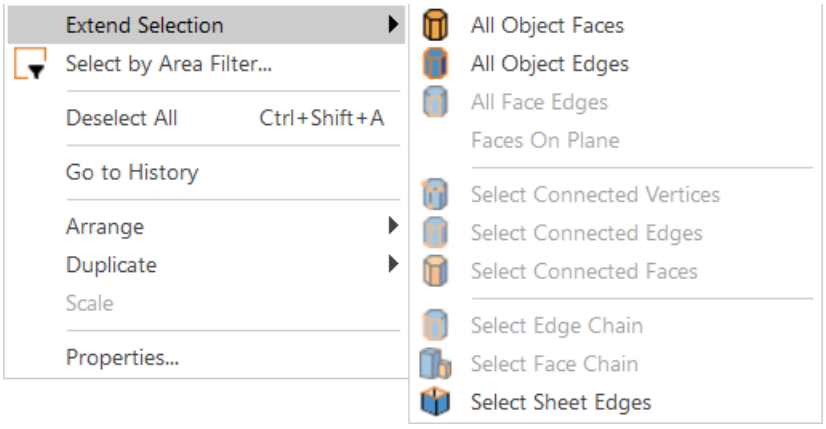
- **Undo** – allows you to undo the last action, which is specified after the word "Undo."
- **Redo** – allows you to redo an undone action, which is specified after the word "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste elements.
- **Delete** – deletes a selected element.
- **Rename**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



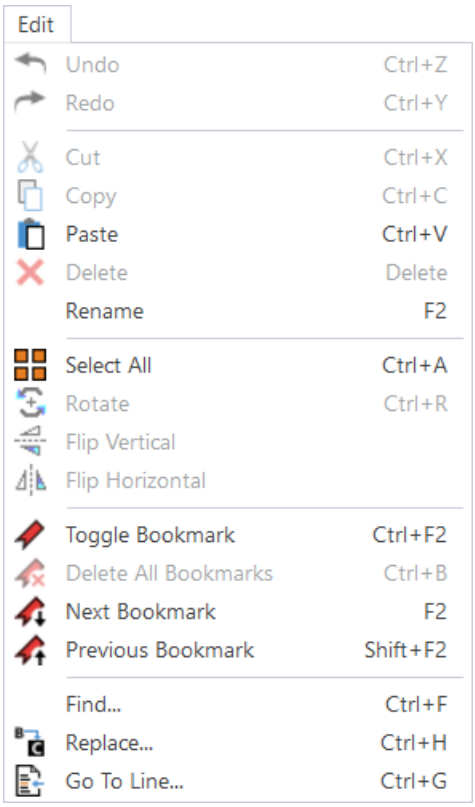
- **Select by Area Filter** – opens the **Select By Area Filters** window.
- **Deselect All** – deselects all selected objects.
- **Go To History** – selects the History Tree entry for the selected object.
- **Arrange** – allows you to move, rotate, or mirror the selected object.

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 Netlist Editor

When the Netlist Editor is the active window, the **Edit** menu appears similar to the following:



The Netlist Editor Edit menu contains the following options:

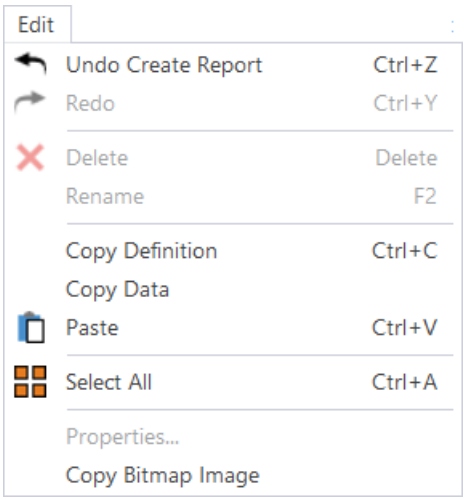
- **Undo** – allows you to undo the last action, which is specified after "Undo."
- **Redo** – allows you to redo an undone action, which is specified after "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste netlist elements.
- **Delete** – deletes a selected element.
- **Rename** – renames a selected element.
- **Select All** – selects the entire netlist.
- **Rotate / Flip Vertical / Flip Horizontal** – deactivated options.
- **Toggle Bookmark** – toggles the display of bookmarks.
- **Delete All Bookmarks** – deletes all bookmarks.

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

- **Next Bookmark / Previous Bookmark** – allows you to navigate between bookmarked lines.
- **Find** – searches the netlist for a string of text.
- **Replace** – searches the netlist for a string of text and replaces it with different text that you specify.
- **Go To Line** – allows you to enter a line number and jump to that line.

Edit Menu for Report Window

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



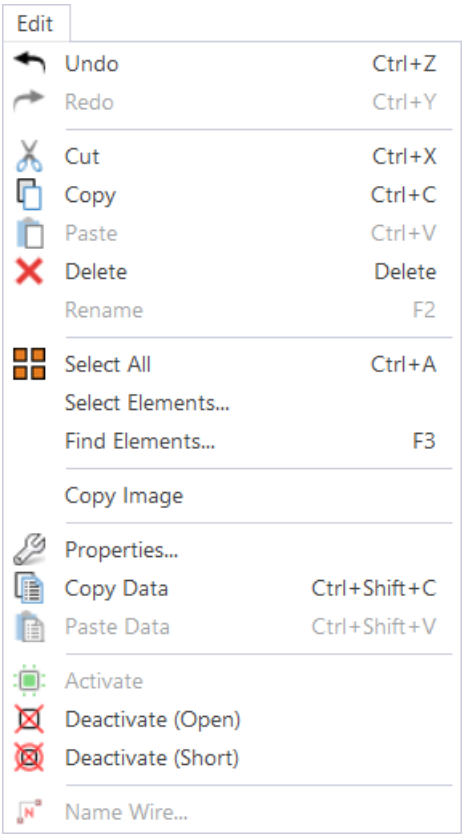
The Report Window Edit menu contains the following options:

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

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

Edit Menu for Schematic Editor

When the Schematic Editor is active in the Design Area, the **Edit** menu appears similar to the following:



The Schematic Editor Edit menu contains the following options:

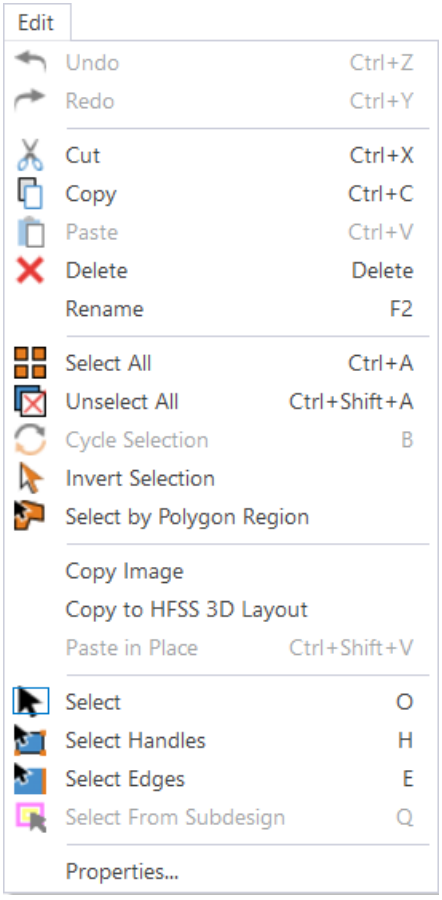
- **Undo** – allows you to undo the last action, which is specified after "Undo."
- **Redo** – allows you to redo an undone action, which is specified after "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste schematic elements.
- **Delete** – deletes a selected element.
- **Rename** – renames a selected element.
- **Select All** – selects everything in the design area.
- **Select Elements** – allows you to select certain nets or components.
- **Find Elements** – opens the **Find Elements** window, where you can search for elements by property name and value.
- **Copy to Clipboard** – copies the visible design area to the clipboard in bitmap format, for pasting into another program or document.

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

- **Properties** – displays the relevant properties window for the selected object.
- **Copy Data** – copies properties data for the selected object.
- **Paste Data** – pastes properties data into a selected object.
- **Activate** – causes the activation of one or more selected components.
- **Deactivate (Open)** – temporarily converts the component into an open circuit.
- **Deactivate (Short)** – temporarily converts the component into a short circuit.
- **Name Wire** – allows you to rename a selected wire.

Edit Menu for Layout Editor

When the Layout Editor is active in the Design Area, the **Edit** menu appears similar to the following:



The Layout Editor 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."
- **Cut / Copy / Paste** – allows you to copy and paste layout elements.

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

- **Delete** – deletes a selected element.
- **Rename** – renames a selected element.
- **Select All / Unselect All** – selects or deselects the entire layout.
- **Cycle Selection** – rotates through a selection of objects.
- **Invert Selection** – selects the opposite of the current selection.
- **Select By Polygon Region** – allows you to draw a polygon in the design area and selects all elements within that polygon.
- **Copy Image** – copies the design area display in bitmap format, for pasting into another program or document.
- **Copy to HFSS 3D Layout** – copies the design into an HFSS 3D Layout and adds that layout to the current project.
- **Paste in Place** – duplicates an object without any X / Y offset or displacement.
- **Select** – puts the cursor in "Select" mode (the default).
- **Select Handles** – puts the cursor in "Select Handles" mode; the cursor will only select handles.
- **Select Edges** – puts the cursor in "Select Edges" mode; the cursor will only select edges.
- **Properties** – opens the relevant Properties window for the selected object.
- **Activate** – causes the activation of one or more selected components.
- **Deactivate (Open)** – temporarily converts the component into an open circuit.
- **Deactivate (Short)** – temporarily converts the component into a short circuit.

View Menu

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

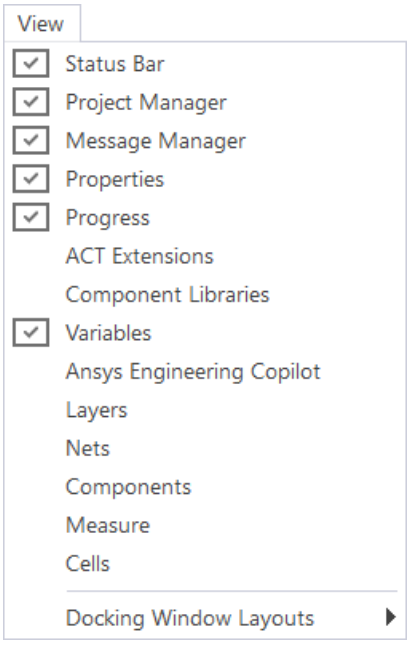
These menus include:

- Basic View Menu
- Layout Editor'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:

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

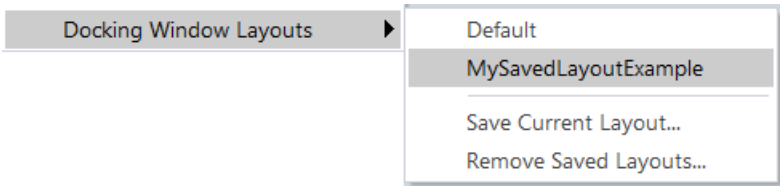


- The check boxes toggle the display of the **Status Bar**, **Message Manager Window**, **Project Manager**, **Properties Window**, and **Progress Window**. You can enable the display of **Component Libraries**, **Layers**, **Nets**, **Components** and **Variables** windows.
- The **ACT Extensions** 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.

- 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.

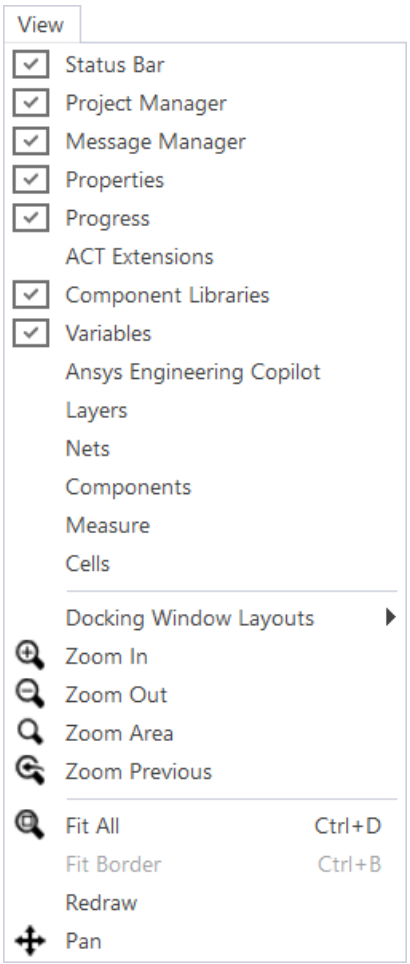
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

Schematic Editor View Menu

When the Schematic Editor is active, the **View** menu appears similar to the following:



In addition to the options available on the Basic View Menu, the Schematic Editor View Menu contains the following:

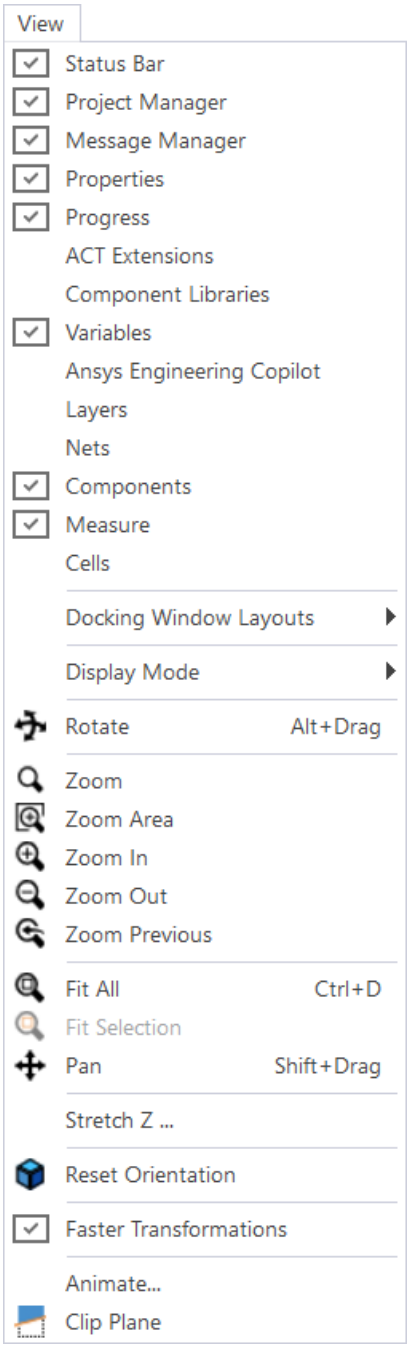
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Zoom In** / **Zoom Out** / **Zoom Area** / **Zoom Previous** – control the portion of the design displayed in the design area.
- **Fit Drawing** – changes the display so that the entire drawing fits to the edges of the design area.
- **Fit Border** – when a border has been added to a drawing, scales the bordered contents to fit within the bounds of the window.
- **Redraw** – redraws the current design.
- **Pan** – allows you to pan across the drawing.

Layout Editor View Menu

When the Layout Editor is active in the design area, the **View** menu appears similar to 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.

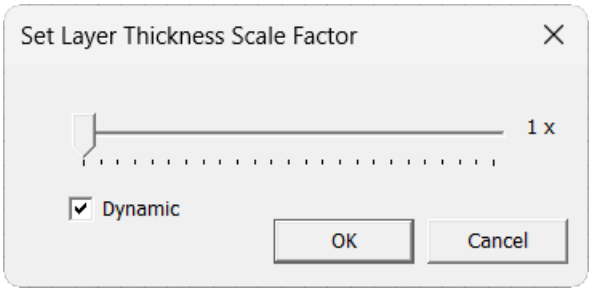


In addition to the options available on the Basic View Menu, the Layout Editor View Menu contains the following:

- **Zoom In** / **Zoom Out** / **Zoom Area** / **Zoom Previous** – control the portion of the design displayed in the design area.

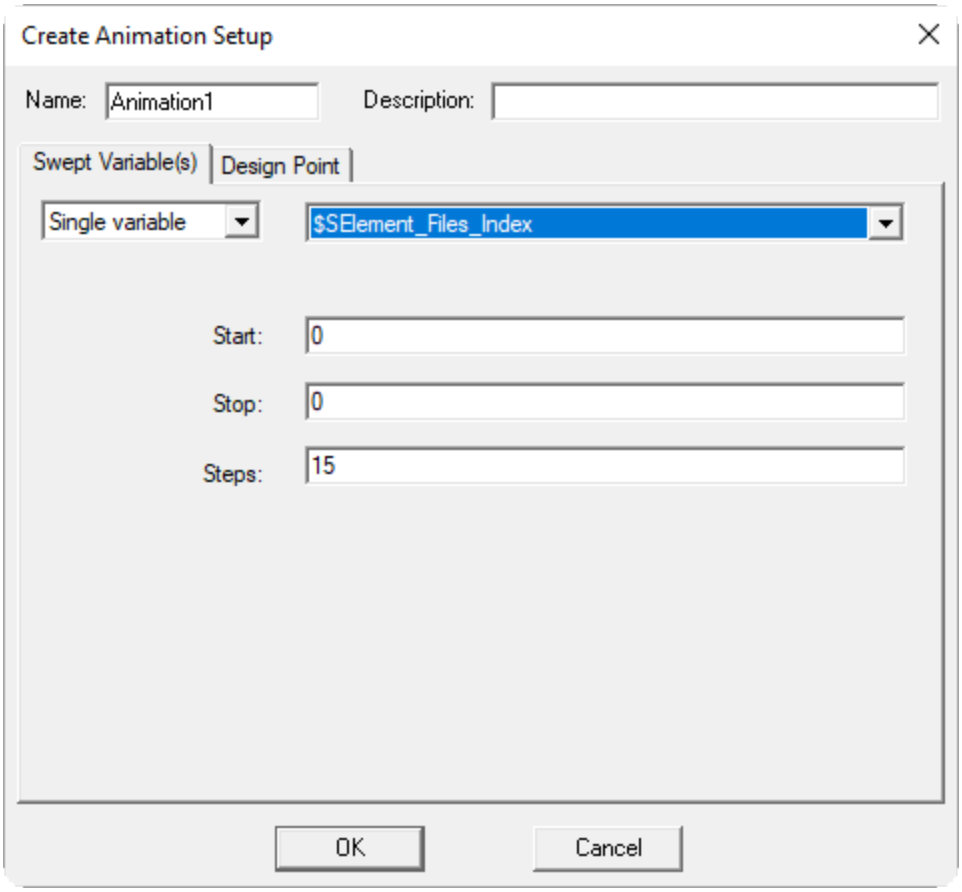
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Fit Drawing** – changes the display so that the entire drawing fits to the edges of the design area.
- **Fit Border** – when a border has been added to a drawing, scales the bordered contents to fit within the bounds of the window.
- **Redraw** – redraws the current design.
- **Pan** – allows you to pan across the drawing.
- **Stretch Z** – opens the **Set Layer Thickness Scale Factor** window, allowing you to change layer thickness.



- **Reset Orientation** – restores the design area to its default view.
- **Faster Transformations** – offers a simplified model during rotate, zoom, and pan operations to speed up performance.
- **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.



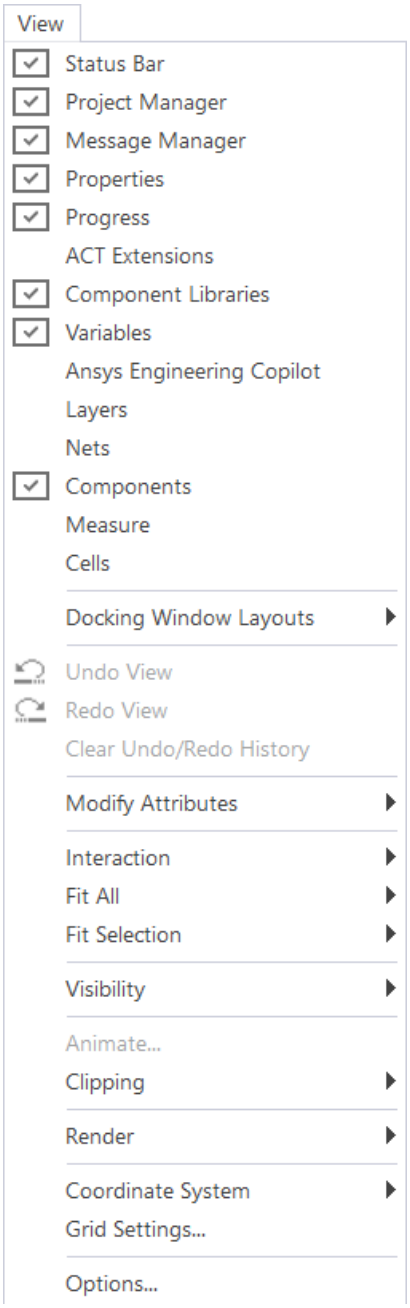
- **Clip Plane** – places a 3D coordinate system manipulator in the design area so that you can define clip planes.

See: Layout Editor.

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.

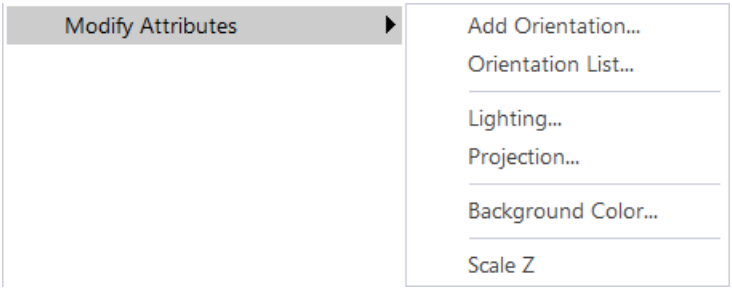


The **View** menu includes the options available on the Basic View Menu. The following additional options are available when a *Modeler* window is active. Note that a number of these options also appear in the View branch of the shortcut menu that appears when you right-click in the Modeler window.

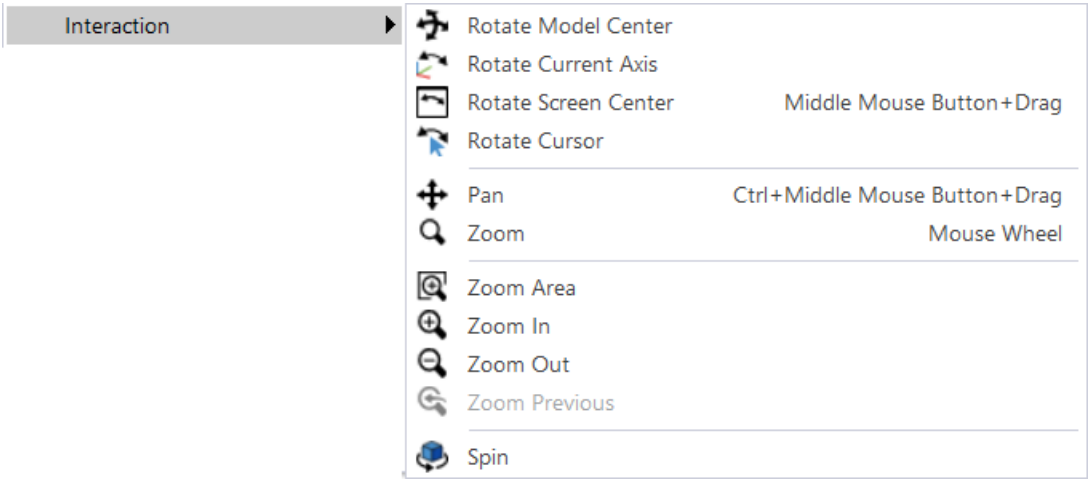
- **Undo View** / **Redo View** / **Clear Undo/Redo History** – allow you to change the view in the design area, based on view history.

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

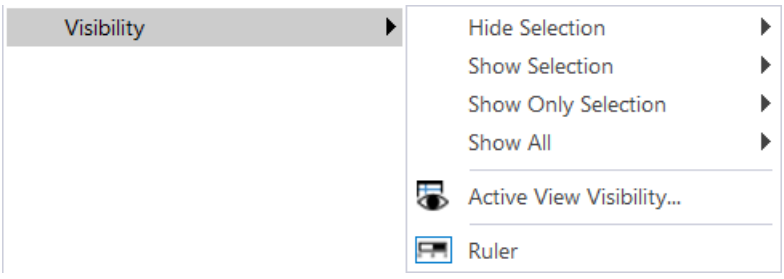
- **Modify Attributes** – offers options to Add Orientation; view the Orientation List; change **Lighting**, **Projection**, or **Background Color**; and **Scale Z**.



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



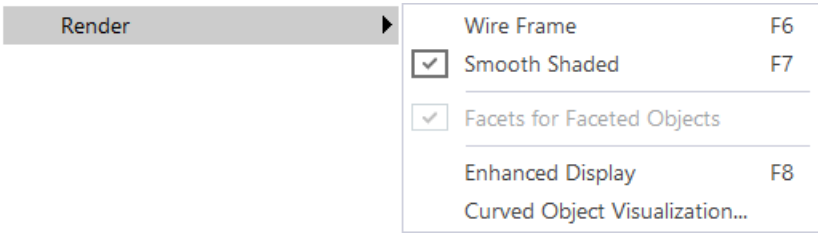
- **Fit All** – allows you to fit all objects to the view window.
- **Fit Selection** – allows you to fit selected objects to the view window.
- **Visibility** – offers options to show and hide selections, objects, shapes, and the ruler.



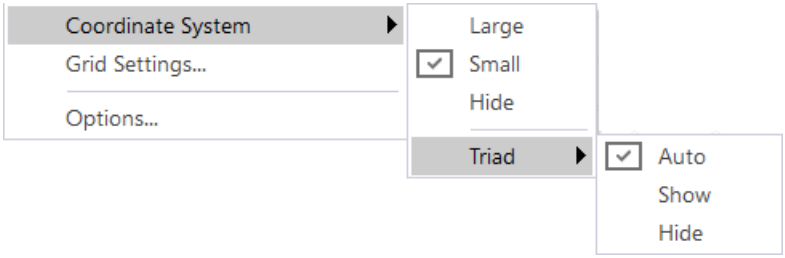
- **Animate** – opens the **Create Animation Setup** window, so that you can begin creating animations.

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.

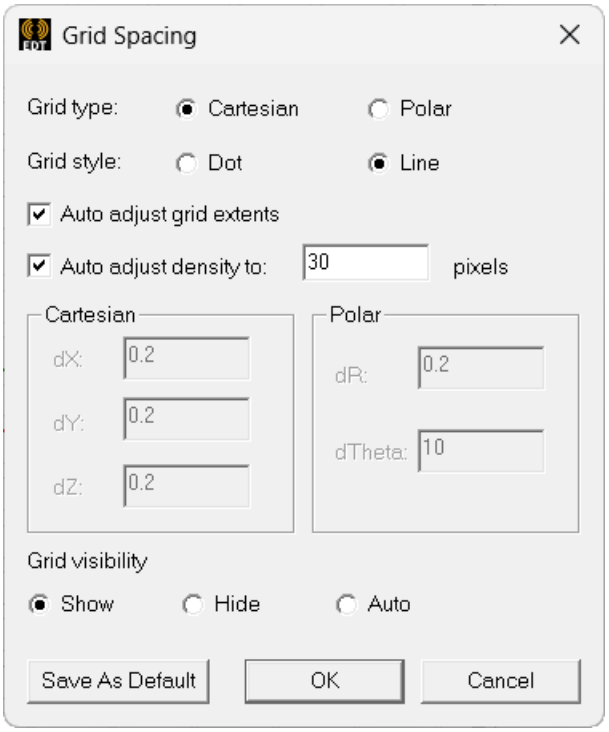


- **Coordinate System** – offers options to change how the coordinate system displays in the design area.



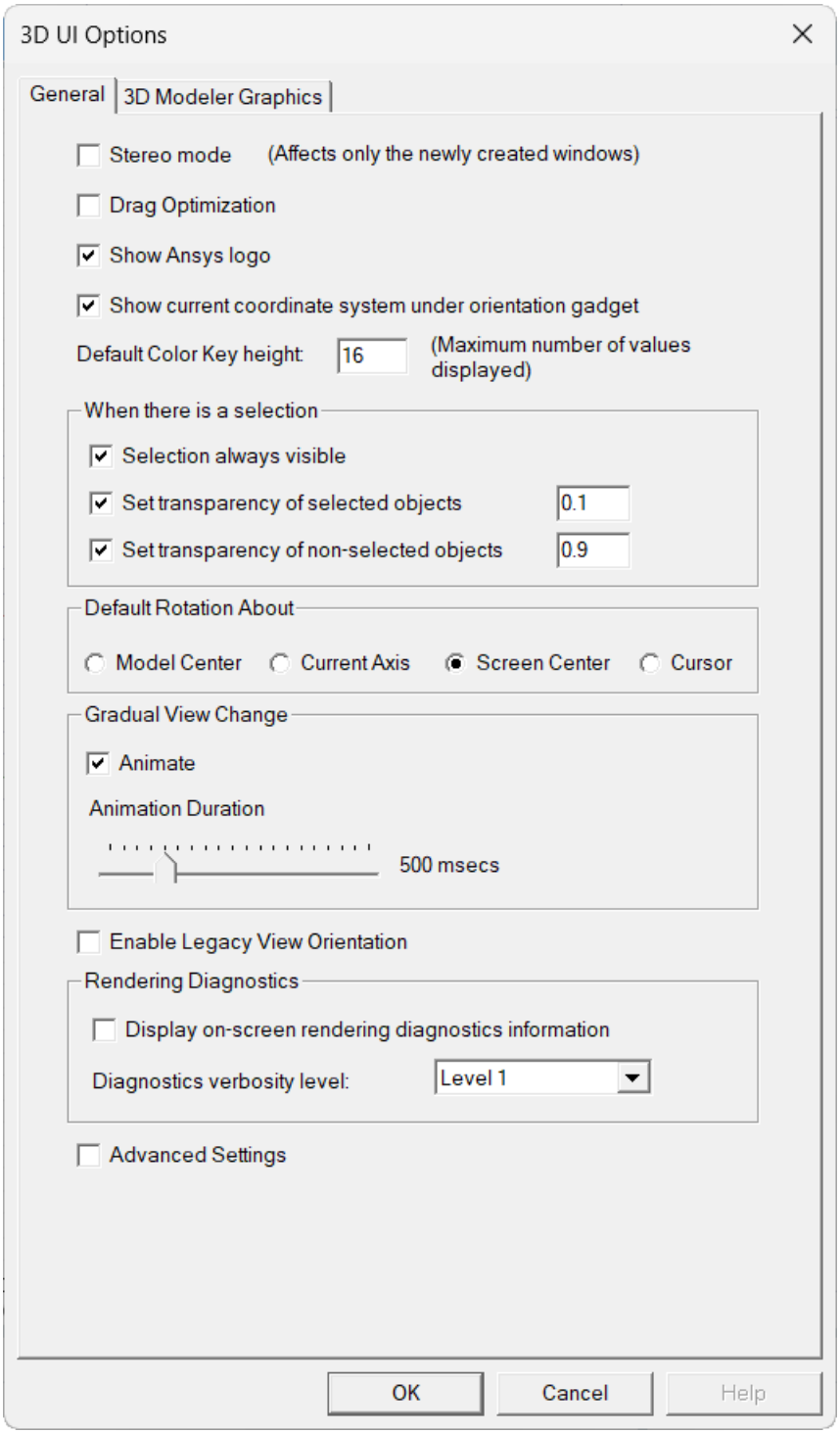
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

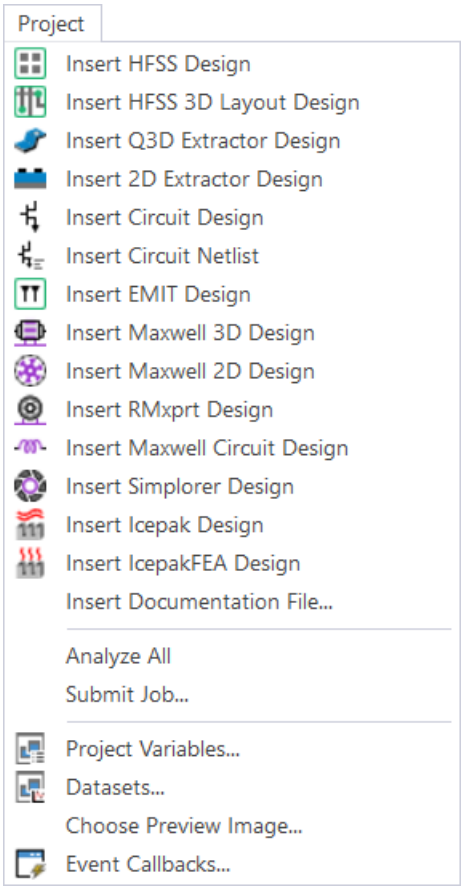
- **Options** – opens the **3D UI Options** window, where you can change additional settings.



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.



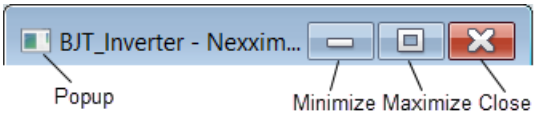
Editor and Design Specific Menus

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.

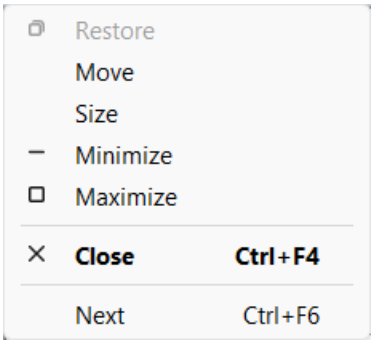
Working with Editor Windows

Each editor window in the Design Area has size controls in its top bar:

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

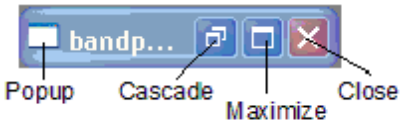


Clicking the rectangle icon on the left opens a pop-up:



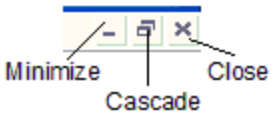
The **Move** and **Size** operations are not needed. To move an editor window, drag it by the top bar into the desired location. To resize an editor window, position the cursor at an edge or corner, and drag the border in or out.

- Click **Minimize** (or use the Minimize icon) to collapse the design window into an icon:



- Clicking the white rectangle on an iconized window opens the same pop-up as the one shown above. To restore an iconized window to its previous size (cascaded or maximized), click the white rectangle, then select **Restore** on the pop-up. Alternatively, use the Cascade or Maximize buttons on the icon to restore the window to the desired configuration. Use the Close button to close the iconized editor window. Reposition the icons anywhere in the Design area by dragging them with the left mouse button. To restore the icons to a neat row at the bottom of the Design Area, use the **Arrange Icons** command on the Window drop-down menu.
- To relocate an iconized window in the Design Area, drag it with the left mouse button.
- Click **Maximize** (or use the Maximize icon) to enlarge the editor window to fill the entire Design Area. In this configuration, Minimize, Cascade, and Close icons appear at the upper right of the Ansys Electronics Desktop, on the same level as the Top Menu bar:

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

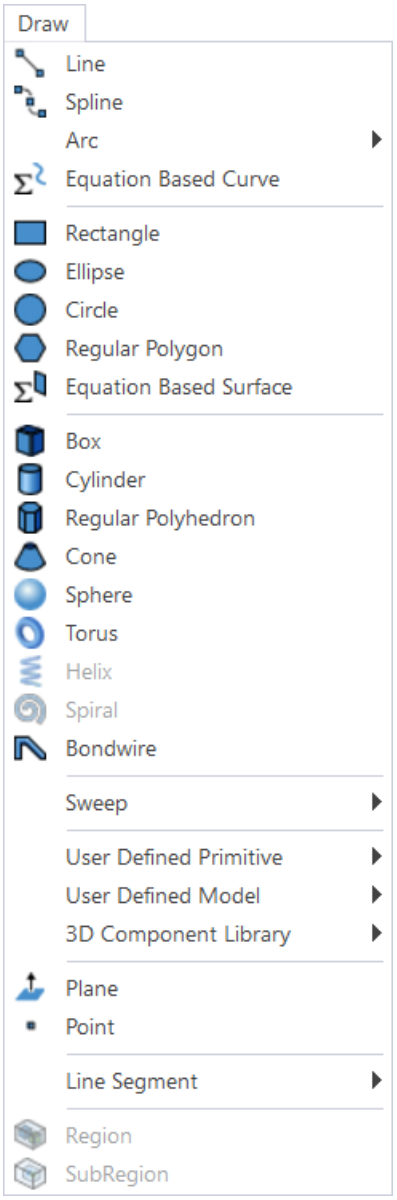


- To restore a maximized window to its initial size and top bar, use the **Cascade** operation.
- Click **Close** (or use the Close icon) to close an editor window.
- Click **Next** to move the active focus to another editor window.
- See Design Area for a summary of the available editor windows.

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.

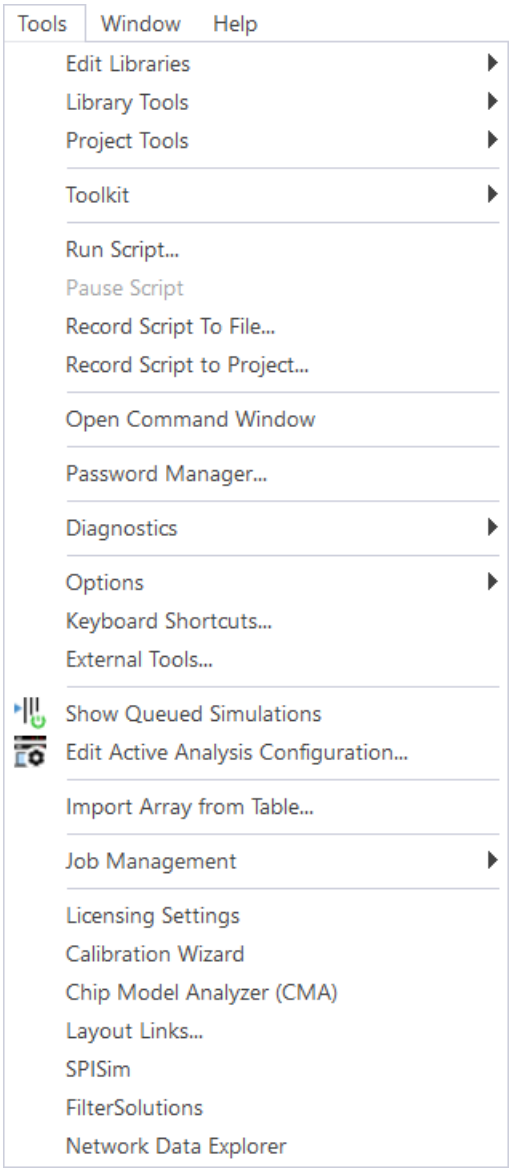
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tools Menu

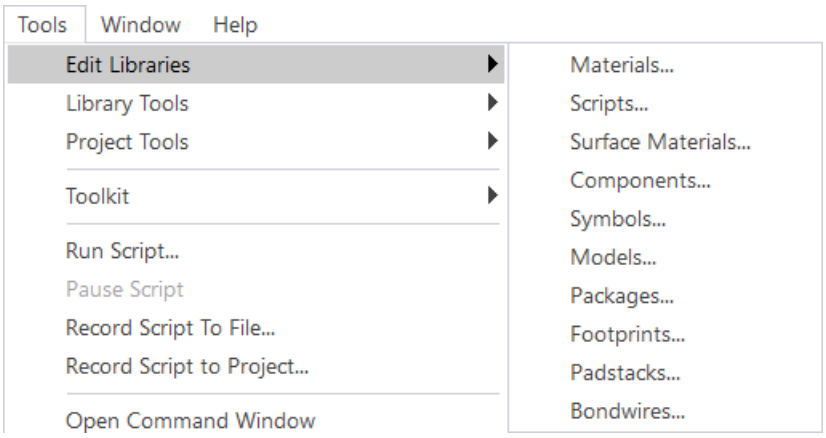
The **Tools** menu contains operations that are common to all design types:

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

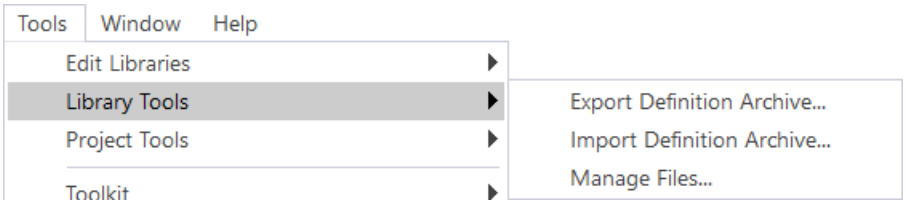


- Click **Tools** > **Edit Libraries** to view commands allowing access to the libraries available for the various Ansys Electronic Desktop solves, such as Materials, Scripts, and Components, Symbols and Models for Simplorer, Icepak, and Circuit.

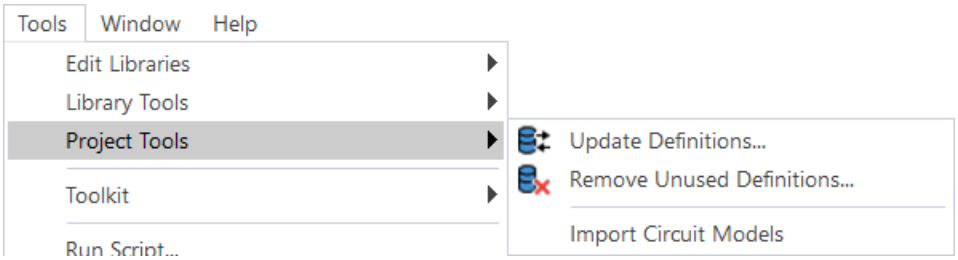
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- 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.



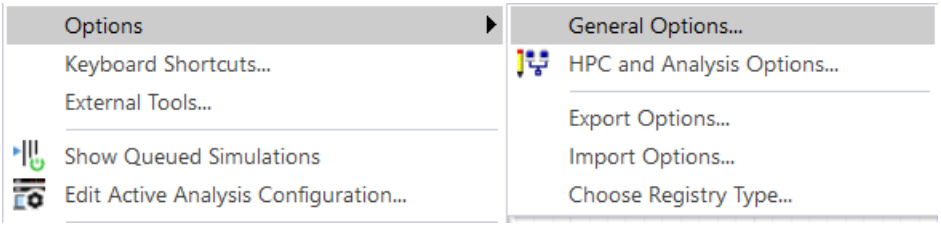
- Click **Tools** > **Project Tools** to manage definition libraries relevant to various solvers.



- 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.

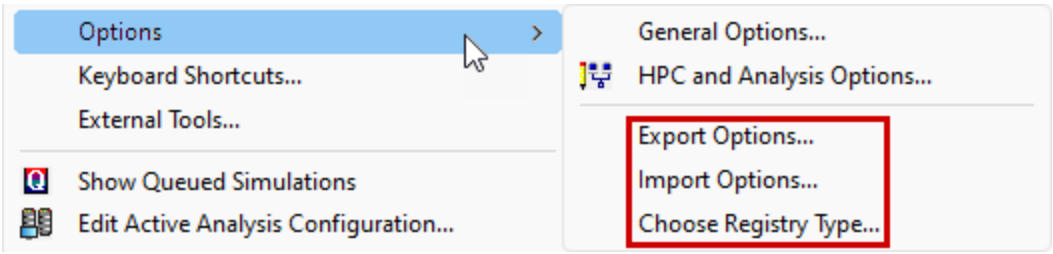
- Click **Tools** > **Keyboard Shortcuts...** to manage your keyboard shortcut behavior.
- Click **Tools** > **External Tools...** to manage menu access to external tools.
- Click **Tools** > **Show Queued Simulations** to view a dialog listing simulations in a queue and providing tools to manage the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

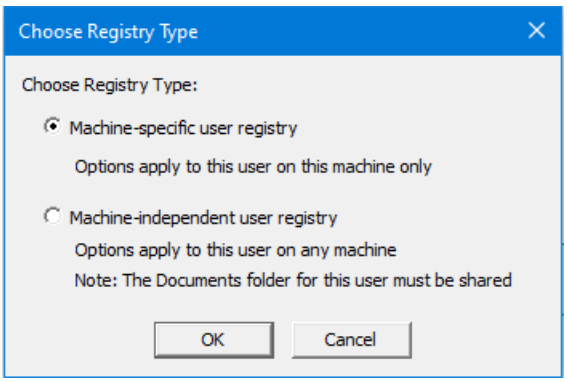
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**

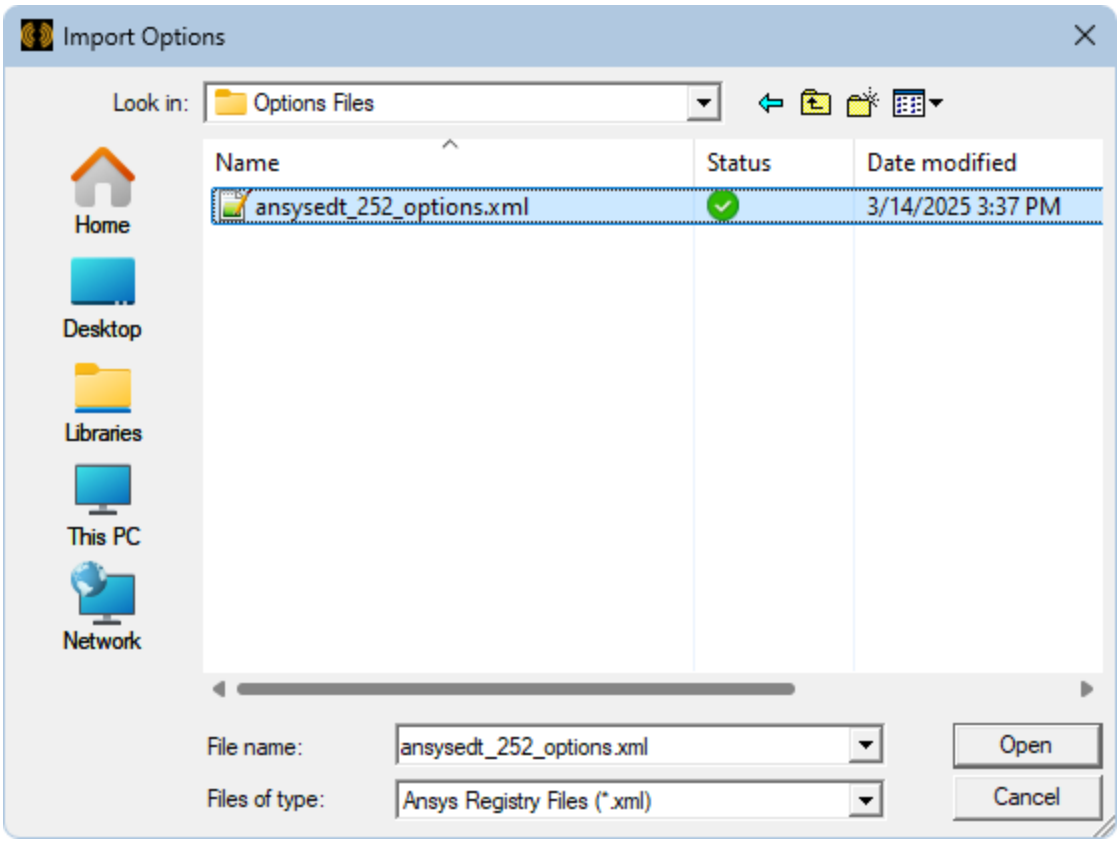
> **Note:**
>
> For a machine-independent user registry to work, the *Documents* folder must be shared by all machines on which you run the software, and you must log into the same account on each machine. The user registry file is stored in the path, ...Documents\Ansoft\ElectronicsDesktop20*yy.m*, where *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:

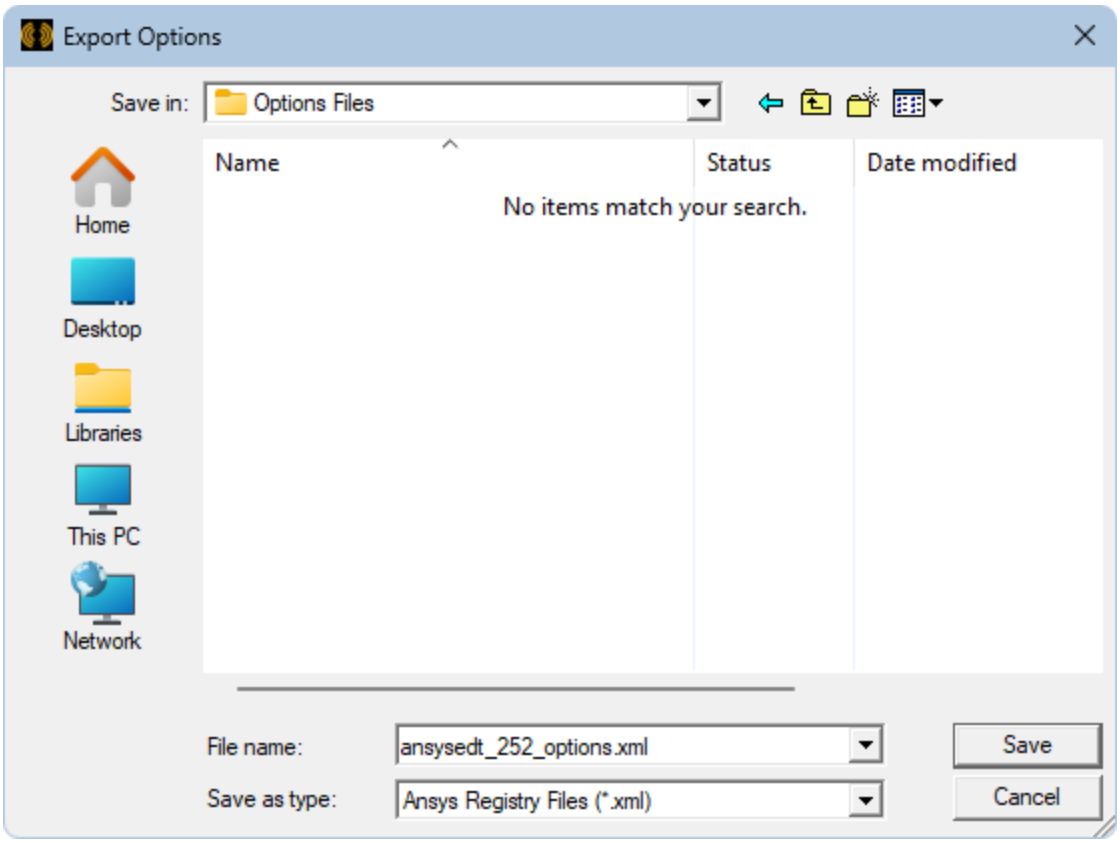
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

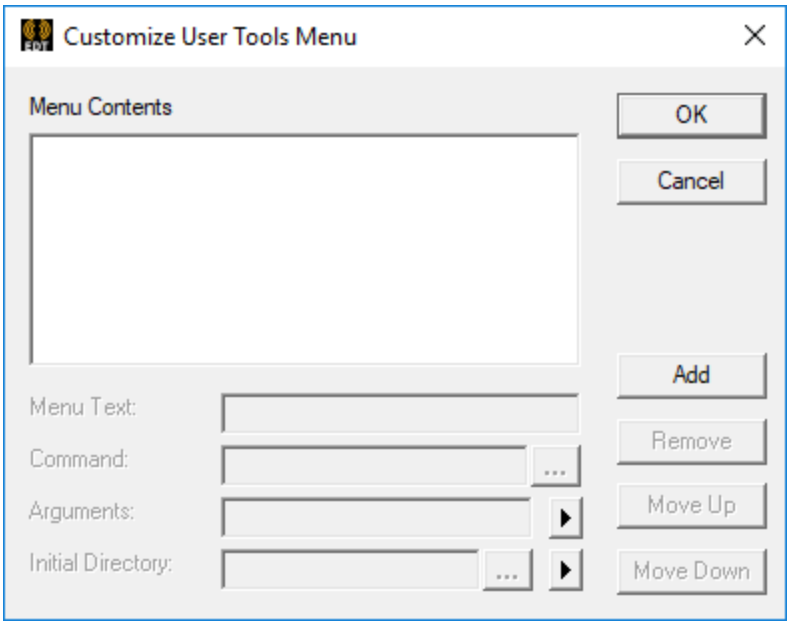
Adding External Tools to the Tools Menu

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.

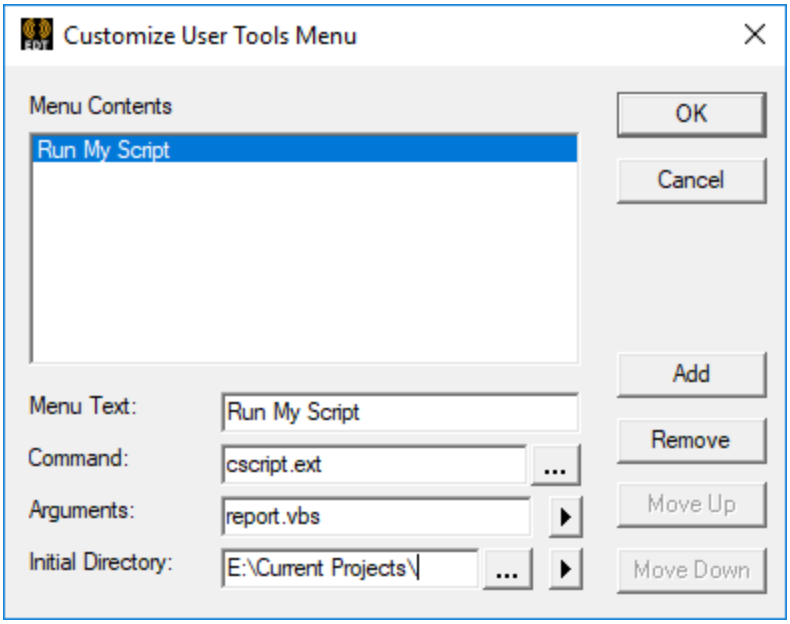
   Arguments field – accepts command arguments from the **>** button menu selections for File Path, File Directory, File Name, File Extension, Project Directory, or Temp Directory.

   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 HFSS 3D Layout 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.

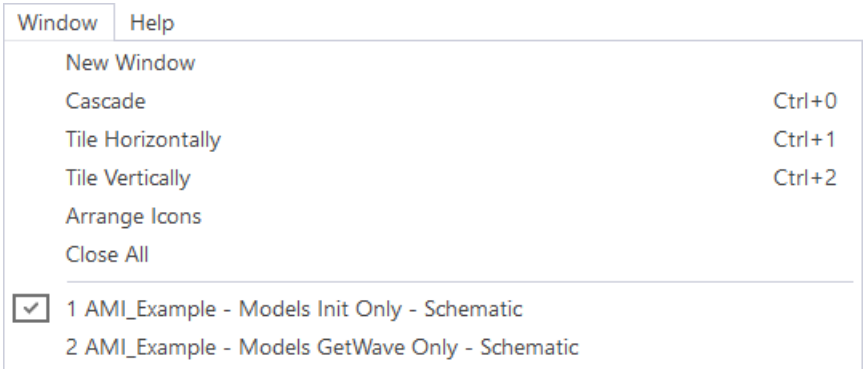
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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).

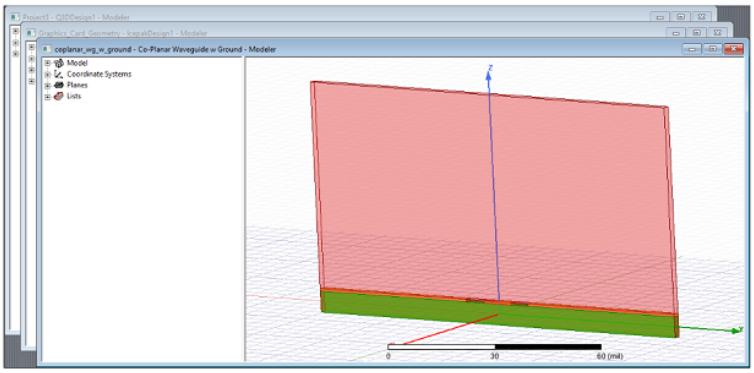
- Click **Tile Horizontally** (or type **h**) to arrange the open design windows in a top-to-bottom sequence:
- Click **Tile Vertically** (or type **v**) to arrange the open design windows in a side-to-side sequence:

> **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.

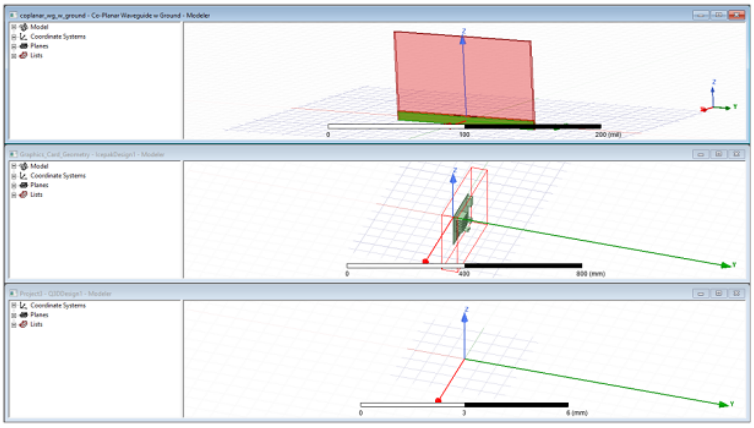
- Click **Arrange Icons** (or type **a**) to restore iconized windows to a neat row at the bottom of the Design Area, after the icons have been repositioned manually.
- Click **Close All** (or type **l**) to close all the editor windows in the Design Area.

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

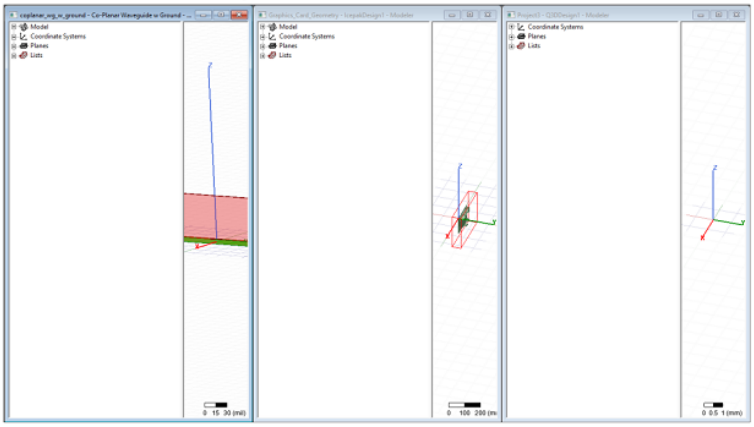
Cascade



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 HFSS 3D Layout are:

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

- **HFSS 3D Layout Help** – opens the HFSS 3D Layout help within the Electronics help system. You can also access PDF versions from within the help system.
- **HFSS 3D Layout Scripting Help** – opens the HFSS 3D Layout 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.

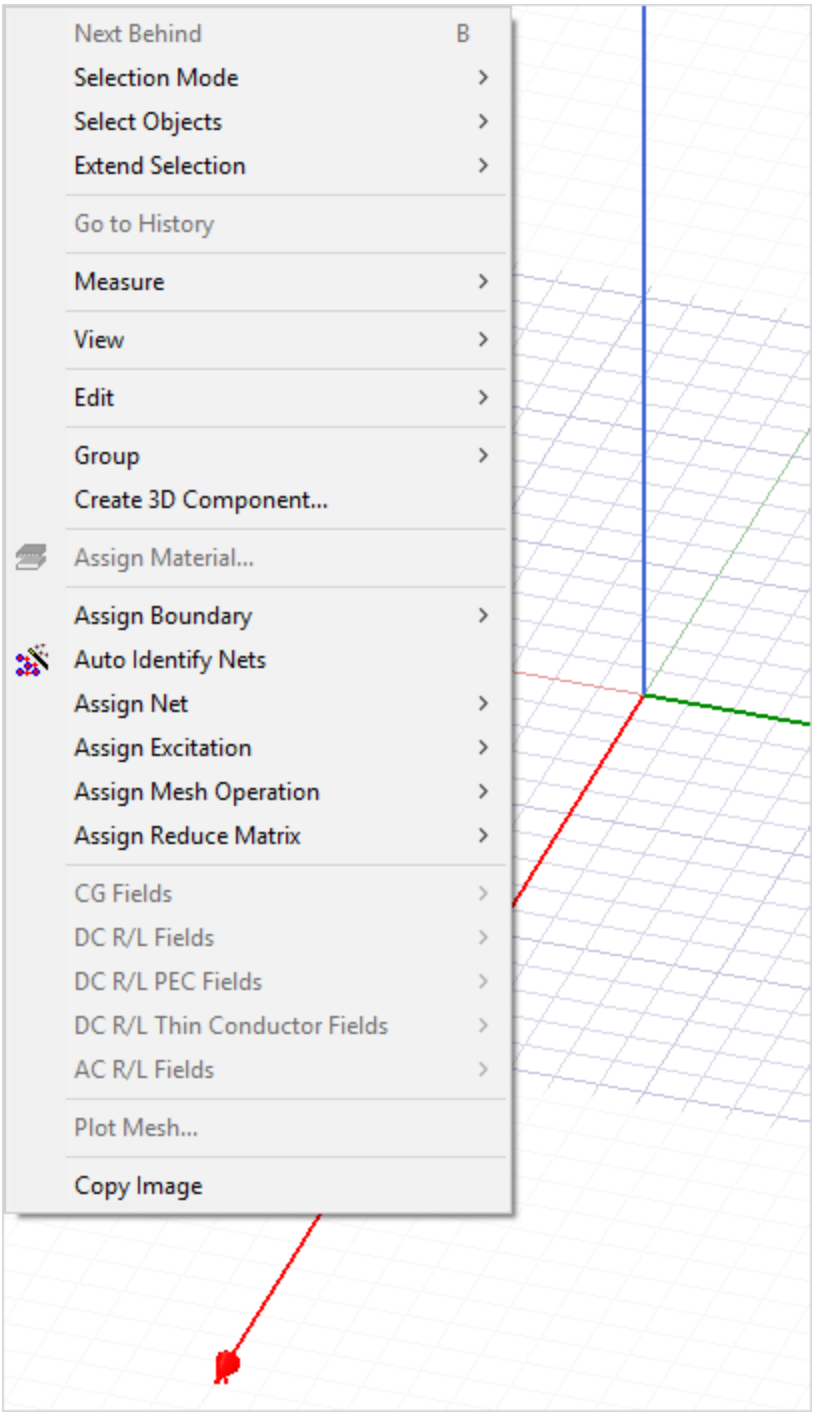
> **Note:**
>
> All commands available on shortcut menus are also available from the menu bar.

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.

Other nodes have shortcut menus appropriate for the context.

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

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
- 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

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

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

Valid XML Elements and Attributes

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

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

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

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

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

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

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

     Child elements of TopMenu:

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

       Child elements of **LeafMenu**:

       - **MenuName** – optional string.

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

         <MenuName>&Tool</MenuName>

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

       - **MenuID** – optional number.

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

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

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

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

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

  Child elements of **CustomMenu**:

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

    Example:

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

    or

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

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

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

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

   Valid values are:

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

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

Note:

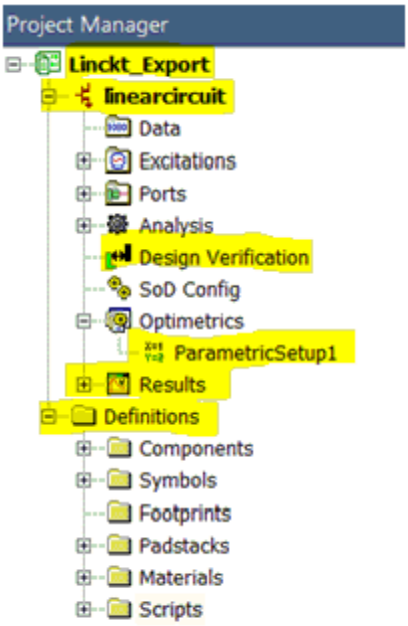
- For “DesignerMenu” and “Context”, only no action setting or “useDefault” is valid.
- The “DesignerMenu” action setting is significant because it is how you specify whether you want to construct your own menus or modify existing default menus. If this is set to “useDefault”, default menu settings are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

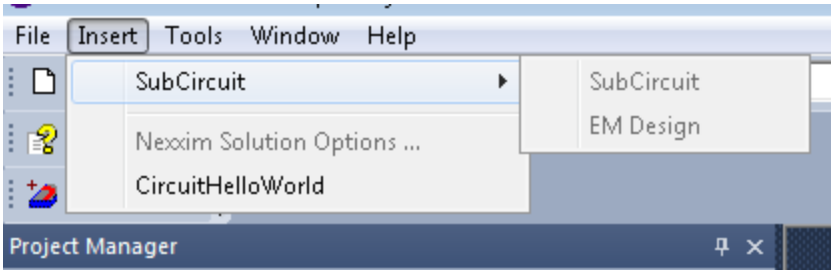
1. **Circuit Netlist.xml Sample**

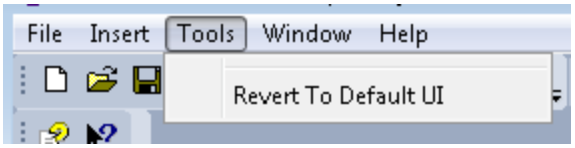
```
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu">
   <Context name="Design">
      <TopMenu>
         <MenuName>Insert</MenuName>
         <pop-upMenu>
            <MenuName>S&amp;ubCircuit</MenuName>
            <LeafMenu>
               <MenuName>&amp;SubCircuit</MenuName>
             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





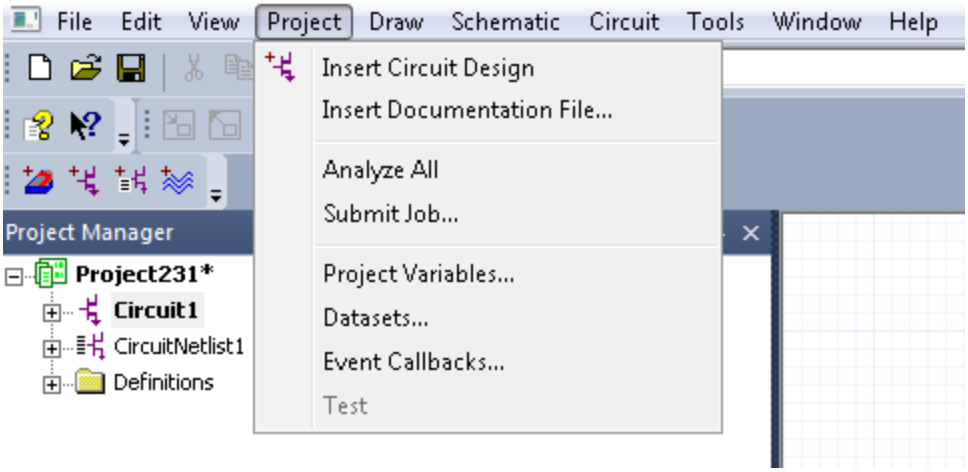
2. **Circuit Design.xml Sample**

```
<?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>
```

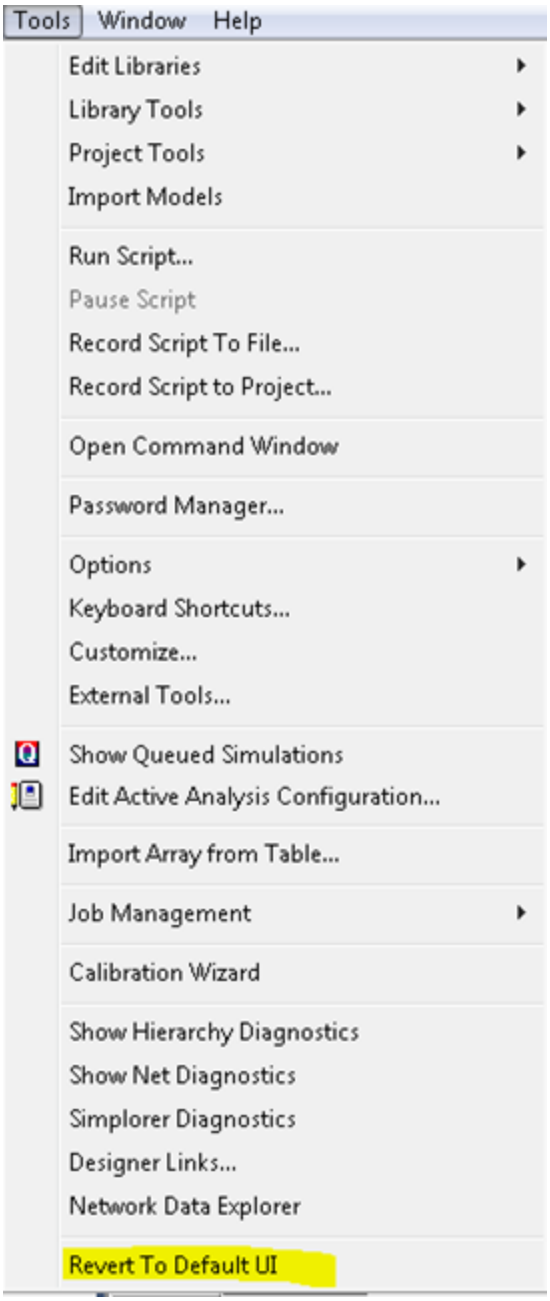

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
            <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.



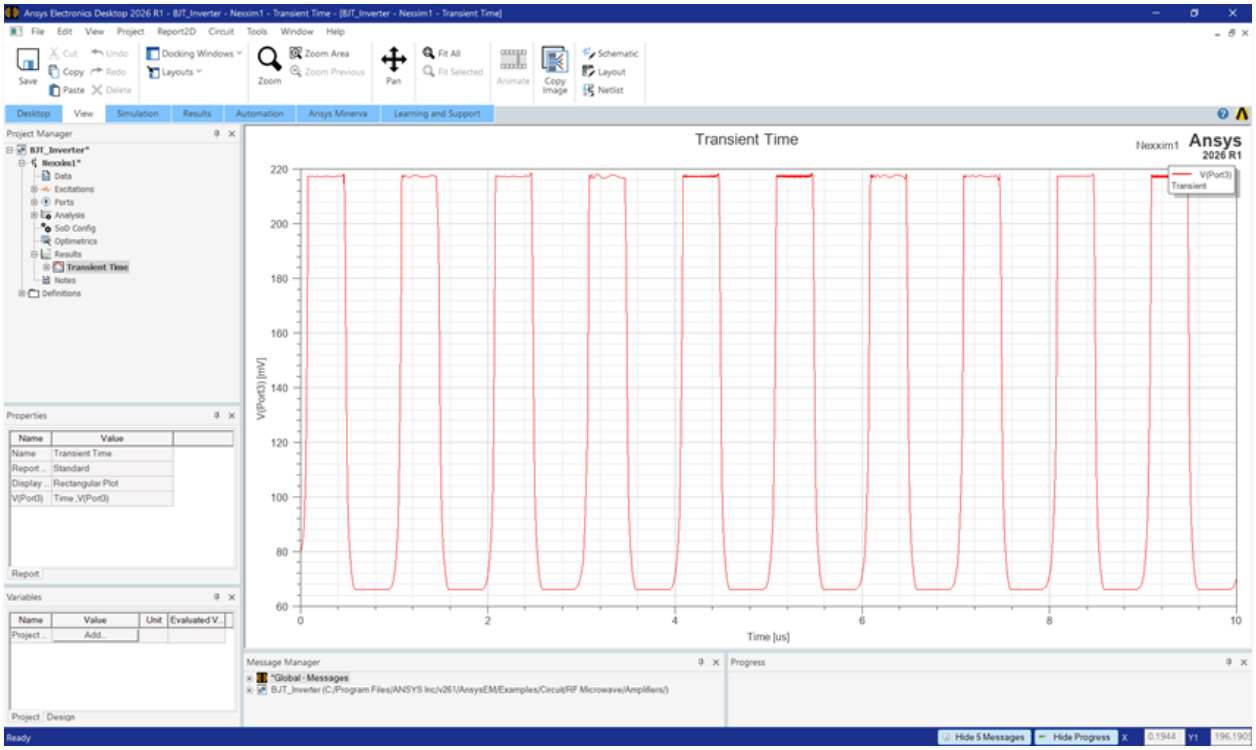
Ansys Electronics Desktop Design Area

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.

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. 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.

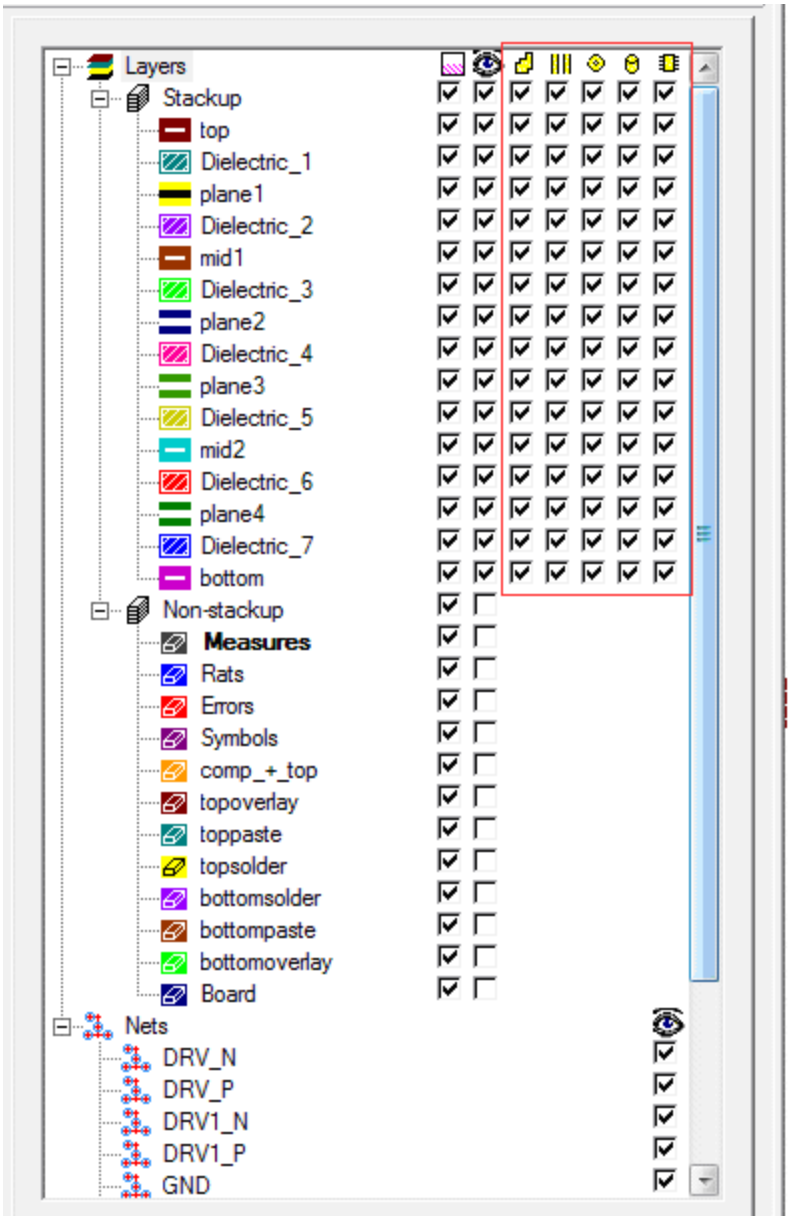
Layout Window

The Layout Window is a dockable Ansys Electronics Desktop window that can be resized and relocated, and can be used to view and configure various layout settings. Use the right-click menu of the Layout Window to configure the following:

- **Set Active** – makes the current layer the active layer.
- **Show This Layer Only** – makes the current layer the only visible layer.
- **Show All Dielectrics** – makes all Dielectric layers visible.
- **Hide All Dielectrics** – makes all Dielectric layers invisible.
- **Show All Signals** – makes all Signal layers visible.
- **Hide All Signals** – makes all Signal layers invisible.

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 **Layout** window to alter the following controls for setting visibility by layout-object type.



The following controls are available:

Controls the visibility of shapes

Controls the visibility of lines (paths)

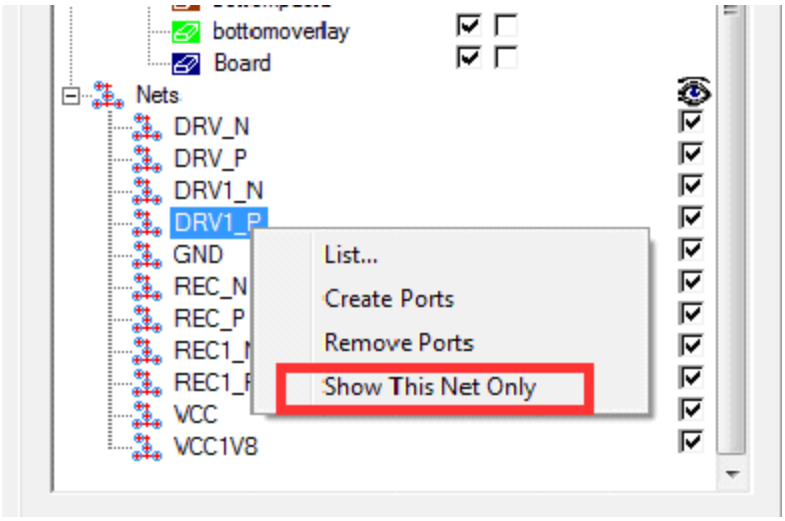
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls the visibility of pads

Controls the visibility of holes

Controls the visibility of components

You can also turn visibility off for all nets but the selected net by right-clicking and selecting **Show This Net Only**.



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.

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

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

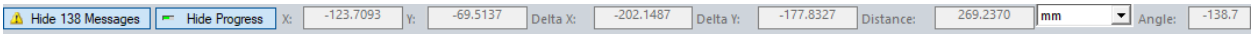
The status bar also displays helpful information about the current selection or command.

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 which is set individually for each design in a project

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



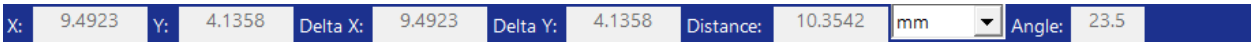
To toggle status bar display:

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

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

Working with the Status Bar and the Layout Editor

The **Status Bar** is located at the bottom of the application window. It displays information about the command currently being performed at lower left, and also displays the following editable values that pertain to the display and positioning of objects selected or drawn in the Layout Editor.



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

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

- X and Y coordinates
- Delta X and Delta Y
- Distance
- Default Units drop-down menu – used to set the unit of measurement individually for each design in a project
- Angle

You can turn off the status bar display by de-selecting the **Status Bar** check box on the Basic View Menu. For more information, see Ansys Electronics Desktop.

Keyboard Shortcuts

The following keyboard shortcuts apply to HFSS 3D Layout 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.

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

Ctrl + N	New
Ctrl + O	Open
Ctrl + P	Print
Ctrl + S	Save
Ctrl + 0	Cascade windows
Ctrl + 1	Tile windows horizontally
Ctrl + 2	Tile windows vertically
Delete	Delete
F1	Open help

Layout Shortcuts

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

The following shortcut key combinations are available when the **Layout** window is active:

Ctrl + A	Select all
Ctrl + Shift + A	Unselect all
Ctrl + C	Copy
Ctrl + D	Fit drawing
Ctrl + "+"	Zoom in
Ctrl + "-"	Zoom out
Ctrl + K	Cross-probe for selected components in schematic editor
Ctrl + M	Align microwave ports
Ctrl + Q	Zoom area
Ctrl + R	Rotate
Ctrl + V	Paste
Ctrl + W	Add connection
Ctrl + X	Cut
Ctrl + Y	Redo
Ctrl + Z	Undo
Delete	Delete
MMB + drag	Rotate (see 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.

Ctrl + MMB + drag	Pan (see note below)
Shift + MMB + drag	Zoom (see note below)

Note:

For view navigation shortcuts (last three rows of the table above):

- **MMB** = Middle Mouse Button.
- These are the default view navigation assignments for 2026 R1. Optionally, you can choose to use the legacy shortcuts that were applicable to 2023 R2 and earlier versions. See Choosing the View Navigation Options for more information.

Report Shortcuts

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

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

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:

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

- To use either the legacy or current view navigation buttons and hotkey combinations, select **Enable Legacy View Navigation**. This option is selected by default.
- To use only the current view navigation behavior and disable legacy button/hotkey support, ensure that **Enable Legacy View Navigation** is cleared.

3. Click **OK**.

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

Current vs. Legacy View Navigation

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

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

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	Wheel or

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

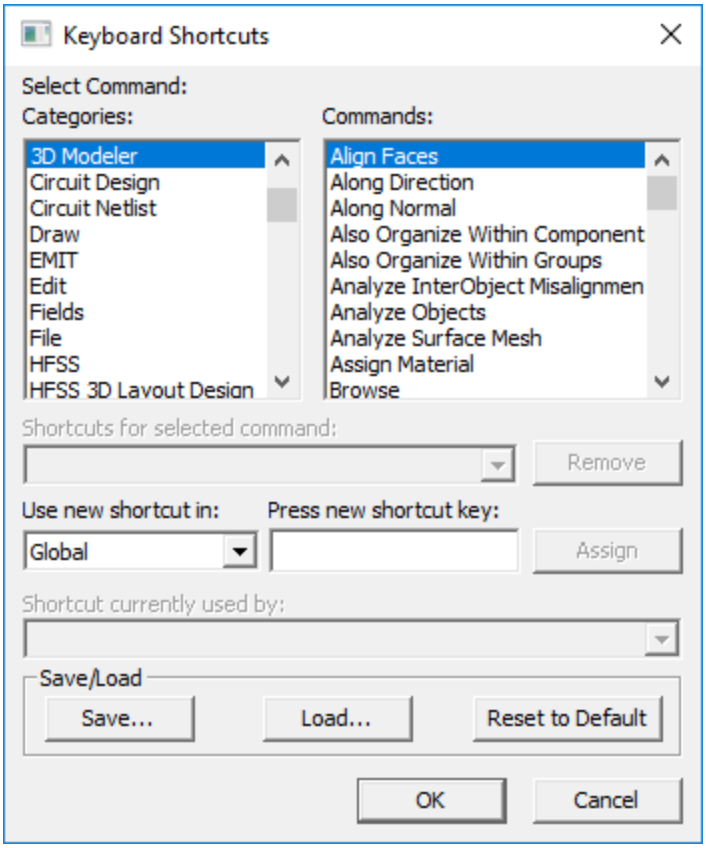
Function	Key / Mouse Button Assignment	
	Current	**Legacy**
	Shift + MMB + drag or Ctrl + "+", Ctrl + "-"	Alt + Shift + LMB + drag or Ctrl + "+", Ctrl + "-"

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

Custom Keyboard Shortcuts

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

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



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

To create a new shortcut key:

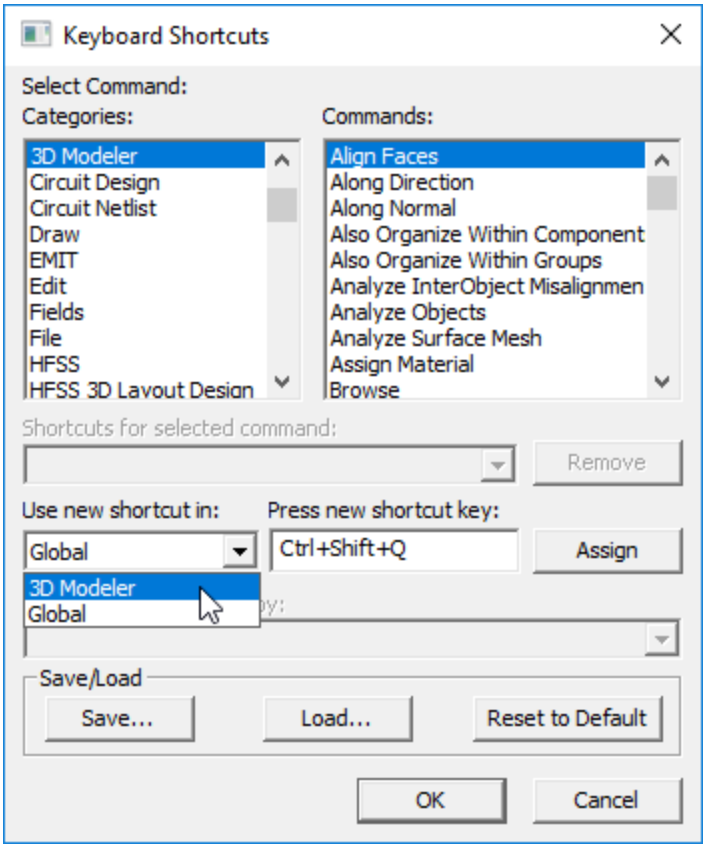
1. Select the **Category** and **Command**.
2. If you want to disable a current assignment for the selected command, click **Remove**.
3. To assign a keyboard shortcut, place the cursor in the **Press new shortcut key** field.

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

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

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

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



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

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

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

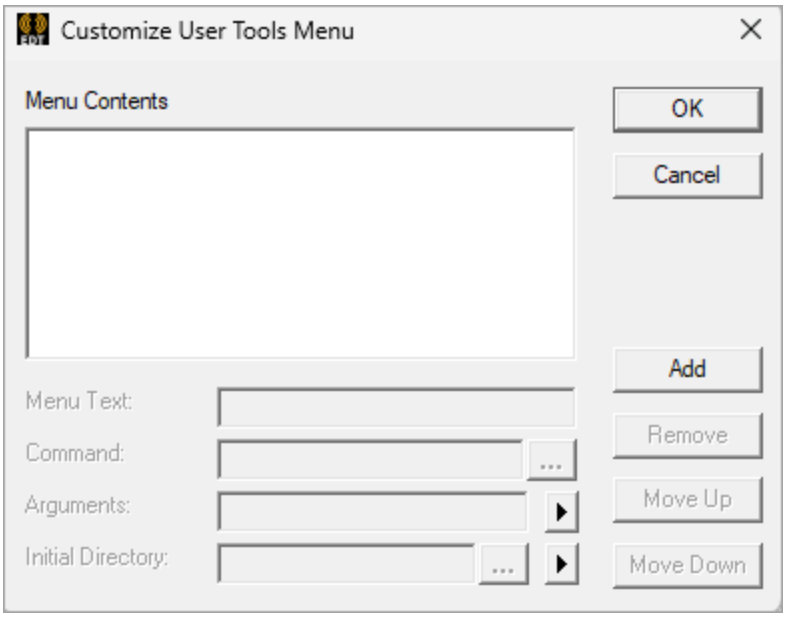
6. Click **OK** to save the current settings, or **Reset to Default**.

Adding External Tools

To add an **ExternalTools** menu to Ansys Electronics Desktop:

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



If a **User Tools** menu has been previously defined, its contents are displayed in the **Menu Contents** window. Navigation buttons allow you to **Add**, **Remove**, **MoveUp** and **MoveDown** items which are displayed in the **Menu Contents** window.

2. Click **Add**. This enables the following fields:

   Menu Text — Displays the text you want to appear in the **UserTools** menu.

   Command – Displays the external executable. Click the browse button to navigate to the file location.

   Arguments – Specifies the argument to be associated with the command. Click the right-arrow button to choose from the following: File Path, File Directory, File Name, File Extension, Project Directory, or Temp Directory.

   Initial Directory – Specifies an initial directory for the command to operate in. Click the browse button to navigate to the file location. Click the right-arrow button to navigate through your desktop or across the network.

3. Click **OK** to add the **ExternalTools** menu to Ansys Electronics Desktop, or click **Cancel** to close the dialog box without 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:

To execute a custom script using the **ExternalTools** customization, enter the name of the program that will execute the script in the **Command** field and enter the location of the script in the **Arguments** field. For example:

For Windows, enter the name of the non-graphical Windows Script Host — 'cscript' — in the **Command** field and enter the location of the script in the **Arguments** field:

Arguments: C:\scripts\myscript.py

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:

   - 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.

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 **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>`

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:

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

- **`-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>`**

  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:

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

- `[designName]` – batch solve all setups for the specified design in the project file.
- `[designName]:Nominal` – batch solve all nominal setups for the specified design in the project file.
- `[designName]:Optimetrics` – batch solve all Optimetrics setups for the specified design in the project file.
- `[designName]:[Nominal/Optimetrics]:[SetupName]` – batch solve the specified Nominal or Optimetrics setup in the specified design.

If you wish to specify whether `-BatchSolve` setups are completed locally or remotely, you can use the following options:

- `-Local` – performs the `-BatchSolve` on the local machine.
- `-Remote -machineList` – performs the `-BatchSolve` on a remote machine. The <*machineList*> should provide a single hostname.
- `-Distributed -machineList` – performs a distributed `-BatchSolve` using a specified machine list.

  The `-machineList` parameter for a `-Distributed` setup can be formatted three ways:

  - `-MachineList list= "<machine1>, <machine2>, ..."` – machine names (either by IP address or hostname) are separated by commas. If the list contains any whitespace, it must be enclosed in quotation marks. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the list. That is, if host1 appears in the list once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

    `list=` accepts the following additional modifiers:

    `<`*`MachineName`*`>:<`*`TasksOnMachine`*`>:<`*`CoresOnMachine`*`>:<`*`RAM Limit in %`*`>:<`*`GPUsOnMachine`*`>`

    Duplicate machine names are not permitted. The integer for `<CoresOnMachine>` must be greater than the integer for `<TasksOnMachine>`. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. 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 host1 and two COM engines will run on host2.

  `file=` accepts the following additional modifiers, in the file itself:

  <MachineName>:<TasksOnMachine>:<CoresOnMachine>:<RAM Limit in %>:<GPUsOnMachine>

  Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. If `-Auto` is specified with a machine list, the number of tasks for each machine must be -1.

  Example:

  ```
  "Orion:4:8:90%:1",
  "Aries:3:12",
  "Pluto:6:12",
  ```

  > **Note:**
  >
  > Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- `-Machinelist num= "<numberofDistributedEngines>"` – This format is used when a scheduler (such as LSF, PBS, SGE or HPC) is used to manage the jobs sent to a cluster of hosts. In a scheduler environment, you can specify the number tasks for distributed processing. In this case, you do not specify the machine names after the flag because the names are provided by the scheduler. For example, in the Windows HPC environment, you can write the number of tasks as follows:

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

```
-Machinelist num=4
```

Distributed setups can also take the following optional arguments. When these are not present, the behavior defaults to single-level distributed solutions with no change in order of precedence among possible distribution types.

The arguments are

- `includetypes= <default>|<distribution type 1, distribution type 2, ...>` – If included distribution types are specified, only the listed distribution types are enabled. If default is specified, the default set of enabled distribution types is used. To see valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  `"includetypes=Frequencies,Mesh Assembly"`
- `excludetypes= <default>|<distribution type 1, distribution type 2, ...>` – If excluded distributed types are specified, all distribution types except those listed will be enabled. If default is specified, the default set of enabled distribution types is used. To see a valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  `"excludelypes=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.

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

The `NumDistributedVariations` option can be used to specify the number of optimetrics variations to solve simultaneously. The default is to solve optimetrics variations sequentially.

Arguments with `-Auto` in a scheduler environment:

- `numcores=`*<total number of cores>*

  Total number of cores, and can be used only with `-Auto` and in a scheduler environment.

- `file=`"*<tmachine list file path>*"

  The specified file can contain line delimited machine specifiers as described above.

- `num=`*<tnum distributed tasks>*

  This is the total number of tasks and used only in a scheduler environment.

- `numgpus=`*<tnumber of GPUs to use>*

  This is the total number of GPUs and used only in a scheduler environment. `numgpus` must be combined with either `num=` or `numcores=`.

  You can also specify how a `-BatchSolve` distributes Optimetrics variations:

- `-Auto` – Without additional parameters, the batch log file will specify that Optimetrics variations be solved sequentially. If `-Auto` is specified with a machine list, the number of tasks for each machine must be -1.
- `-Auto [NumDistributedVariations=<num>]` – You can specify an integer value greater than 1. This is the number of variations that will be solved in parallel.

- **`-BatchExtract <BatchExtract script file name> <project file name>`** – allows the following commands to be executed non-graphically via script and without checking out any GUI licenses:
  - `ExportProfile`
  - `ExportConvergence`
  - `ExportMeshStats`
  - `ExportNetworkData`
  - `ExportNMFData`
  - `ExportEigenmodes`
  - `ExportTransientData`
  - `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.

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

> **Important:**
>
> - `-Ng` must be used with `-BatchExtract` or it will fail.
> - Only the scripts listed above are supported for `-BatchExtract`. Including unsupported script commands will terminate script execution.

- **`-RunScript <script file name>`** – runs the specified script. You can use the -Scriptargs option to add one or more arguments to this command, and the -Iconic option.
- **`-RunScriptAndExit <script file name>`** – runs the specified script and exits Electronics Desktop. You can use the -Scriptargs option to add one or more arguments to this command. You can also use -Iconic, -LogFile, and -WaitForLicense.

> **Note:**
>
> `-BatchSolve <DesignName>` is mutually exclusive with `-RunScriptAndExit <ScriptName>`.

- `-Monitor` – during non-graphical analysis, you can monitor progress and messages. Progress, warning and info messages are logged to the standard output stream. Error and fatal messages are logged to the standard error stream. Schedulers intercept these streams and provide commands for display of this output. See individual scheduler documentation for specifics.
- `-grpcsrv` – For external cpython scripting support. If not specified with **ansysedt.exe**, the Start GRPV server listens on a port in the default range 50051:51051. If specified, Start GRPV server listening on the the specified port number. An error is issued if the port is already used. For example:

```
ansysedt.exe // Start GRPV server listening on a port in the
default range 50051:51051.
ansysedt.exe -grpcsrv portnumber // Start GRPV server listening
on the the port number. error if the port already used.
ansysedt.exe -grpcsrv 50051:50150 // Start GRPV server
listening on a port range 50051:50150.
ansysedt.exe -grpcsrv 50051:100 // Start GRPV server listening
on a port range 50051:50151.
```

Job Management from the Command Line

- `-showmonitorjob` – Launch the monitor job dialog.
- `-showsubmitjob` – Launch the submit job dialog.
- `-showselectscheduler` – Launch the select scheduler dialog.

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

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:

```
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:

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

- `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.

  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.

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

- `-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.
- `-validateonly` – If true, validation will be performed, but none of the setup will be solved. Allowed values: true, "= 1", false, "= 0".
- `-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.

  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

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. 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.

> **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

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 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
"'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'
```


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 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

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

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

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 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

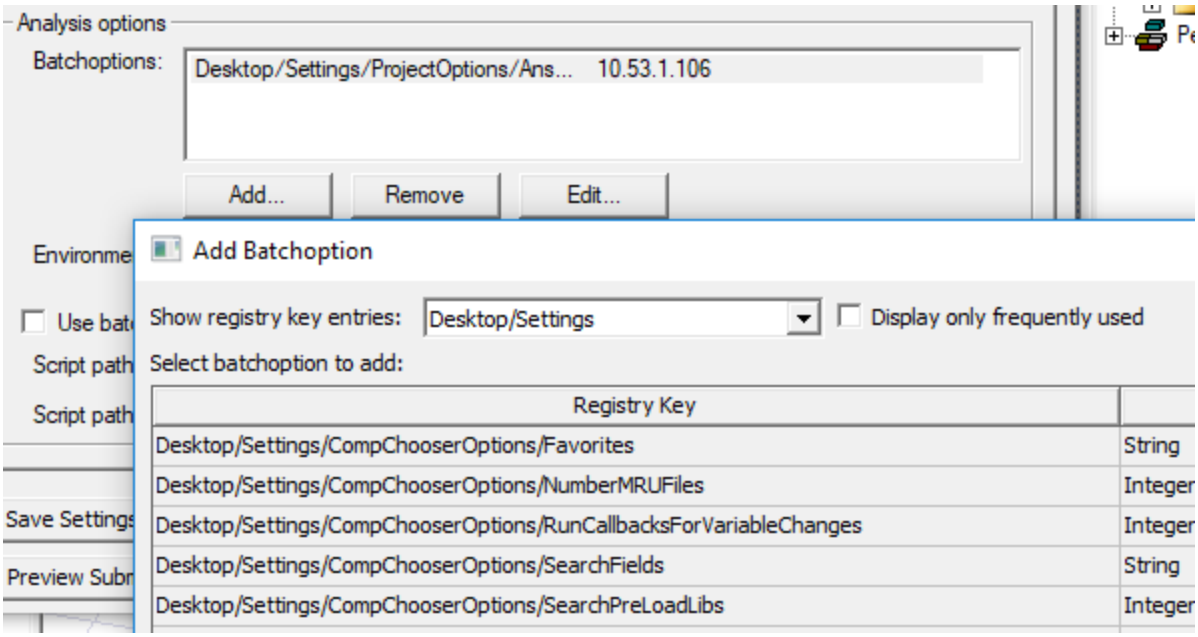
Registry Key	Default Value	Units or Values	Description
TempDirectory	Set by installer	-	Directory for temporary files

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

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 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***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

    Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. 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

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

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***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. 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:

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

- **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.

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.

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

- **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"
```

Specifying Project Files

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

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

You can specify an archive file instead of a project file. If **-Batchsolve** is set, the project will be automatically restored and solved. Otherwise, you are prompted for a restore location, and the project will be restored and opened.

When a **-Batchsolve** is being performed on an archive file, you may also specify **-archiveoptions**:

- **overwritefiles** – allows non-project/results-extracted files to overwrite existing files.
- **path= <***projectFilepath***>** – extracts the project file and associated files to the specified path. If not specified, the archive will be extracted into the same directory as the archive file.
- **repackageresults** – Add batchsolve results back to archive file.
- **winpath= <***windowsProjectFilepath***>** – specifies the Windows-specific path to the extracted project file. This is used when a batch job is to be run on a Linux system, but monitored on Windows.

Options

The following options can be associated with one or more of the run commands:

- **-autoextract** – exports profile (as text), convergence (as text), and report data (as CSV) for the requested project/design/setup in a batch job. Once the solve is complete, an export directory is created (for example, "Project1.aedtexport" for a project named "Project1.aedt") that contains a sub-directory for each design name. You can also specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file (for import and display in Ensight) for all valid field plots. Export files reside within each design-name directory, and include setup name, design variation, job ID, and problem type, as applicable.

  > **Note:**
  >
  > - The -autoextract option is only valid when used with -BatchSolve
  > - The -autoextract option is automatically added for all Ansys Cloud 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.

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

- **-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.
- **-validateonly** – If true, validation will be performed, but none of the setup will be solved. Allowed values: true, "= 1", false, "= 0".
- **-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:

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

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 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.

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

- 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",
    "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_
```


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

```
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":
        [
            {
                "machine": "sjohpcw2k16fe",
                "numTasks": -1,
                "numCores": 1,
                "RAMPercent": 90,
                "numGPUs": 1
            },
            {
                "machine": "dcuww10core01",
                "RAMPercent": 80
            }
        ]
    },
    "monitor": true,
    "ng": true,
    "runscript": "\\\\dataserver\\home\\user\\script.py",
```


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

```
    "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 HFSS 3D Layout 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.

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.

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 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*.

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.

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

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:

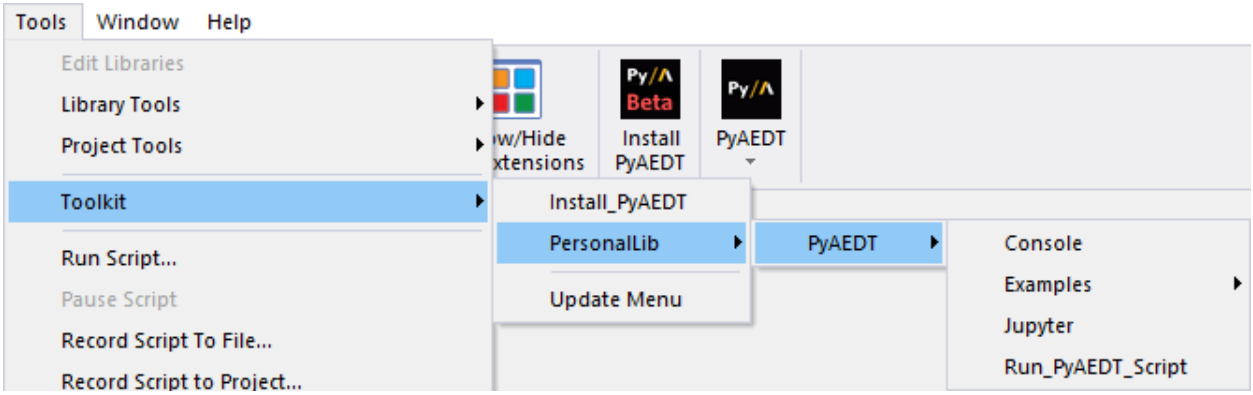
- **Console** – launches the PyAEDT console.
- **Jupyter Notebook** – launches Jupyter Notebook (a computational notebook) in an internet browser.
- **Run PyAEDT Script** – launches a file browser allowing you to select a Python script to run via PyAEDT.

To install PyAEDT:

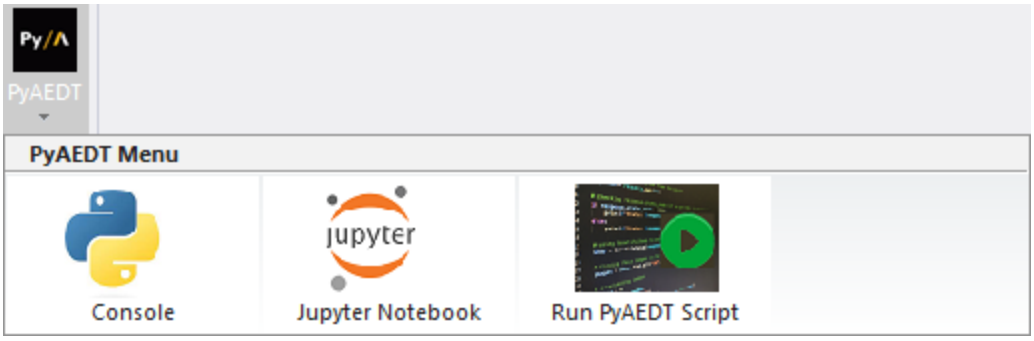
- From the **Automation** tab, click **Install PyAEDT**.

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

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



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



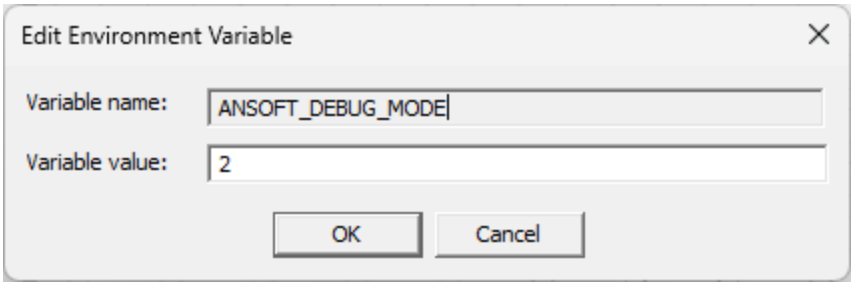
Debug Logging

Electronics Desktop provides error logging capabilities.

To begin logging errors:

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

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



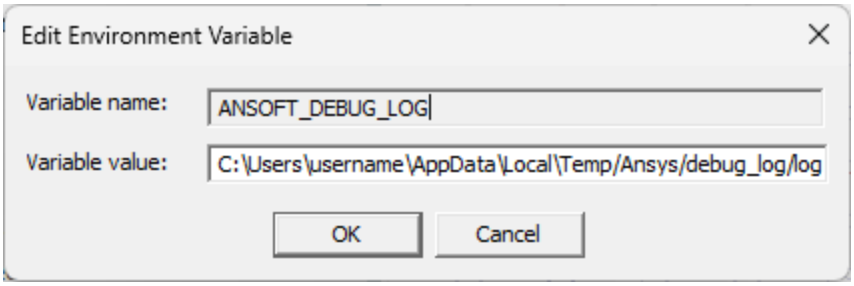
2. Enter a **Variable value**.

   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.

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



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.

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.

> **Note:**
>
> Not all options and capabilities documented in this section apply to all design types.

Ansys Electronics Desktop Files

When you create any project in Ansys Electronics Desktop, it is given an .aedt file extension and stored in the directory you specify. For Legacy projects, the transition is handled as follows:

- When opening a legacy project in interactive mode, a dialog box appears, informing you that the legacy file extension is no longer supported and that if you continue, the project will be converted to the new .aedt extension. If you agree, the project and results directory are renamed/moved to the new file extensions immediately. The read continues with the

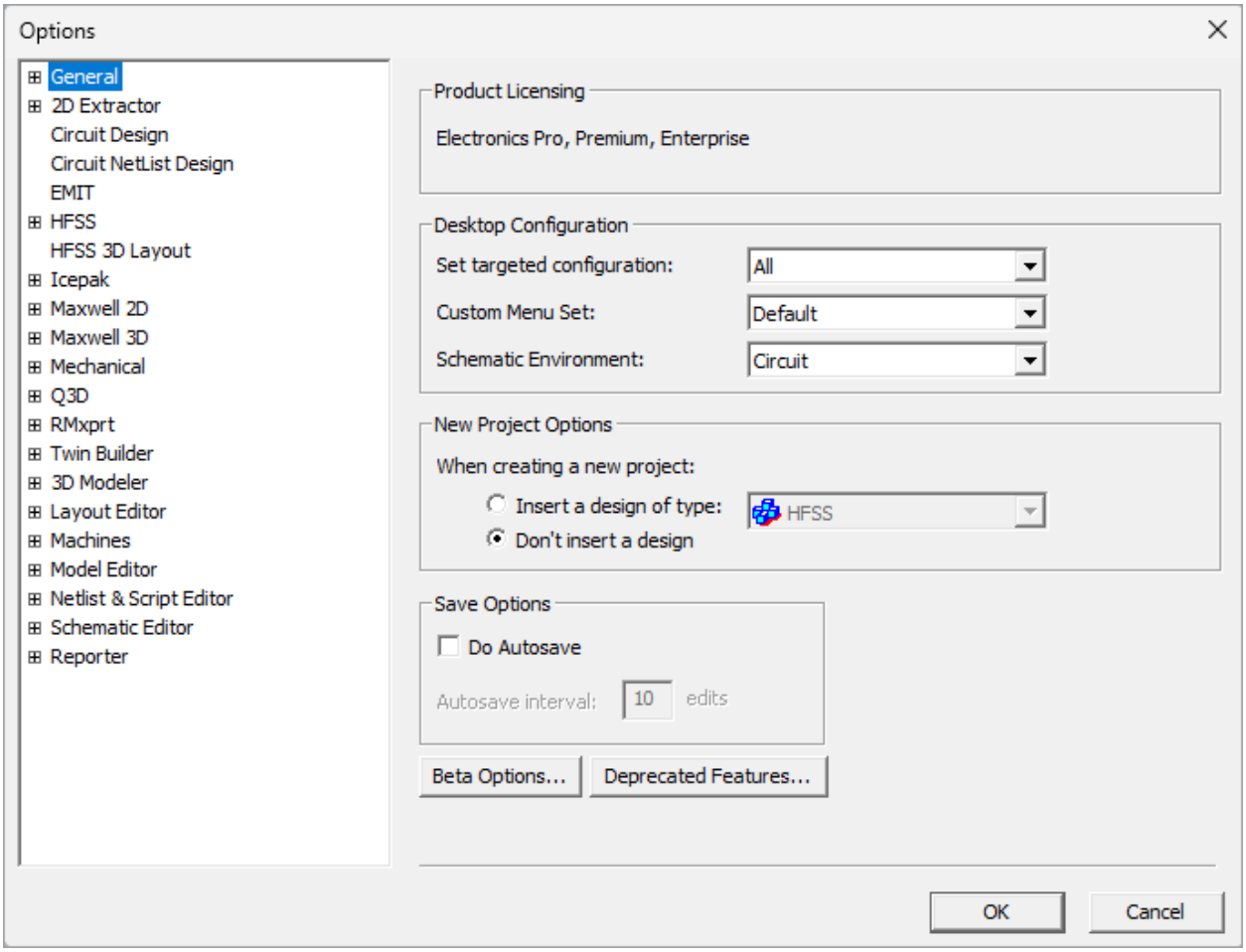
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
- Attempts to run batch solve or non-graphical with a legacy project return an error. There is no automatic/hidden conversion of file extensions. Note that the existing BatchSave command can be used to convert many projects to the new extension and version.
- Workbench integration has not been modified. It continues to open legacy projects without a warning and copies results from legacy to the new extension.

Setting Project Options

Tools > **Options** > **General Options** opens the **Options** window.



The left pane provides access to the following sets of options:

- General
- 2D Extractor

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

- Circuit Design
- IcepakFEA
- RMxprt
- Twin Builder
- 3D Modeler
- Machines
- Model 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.

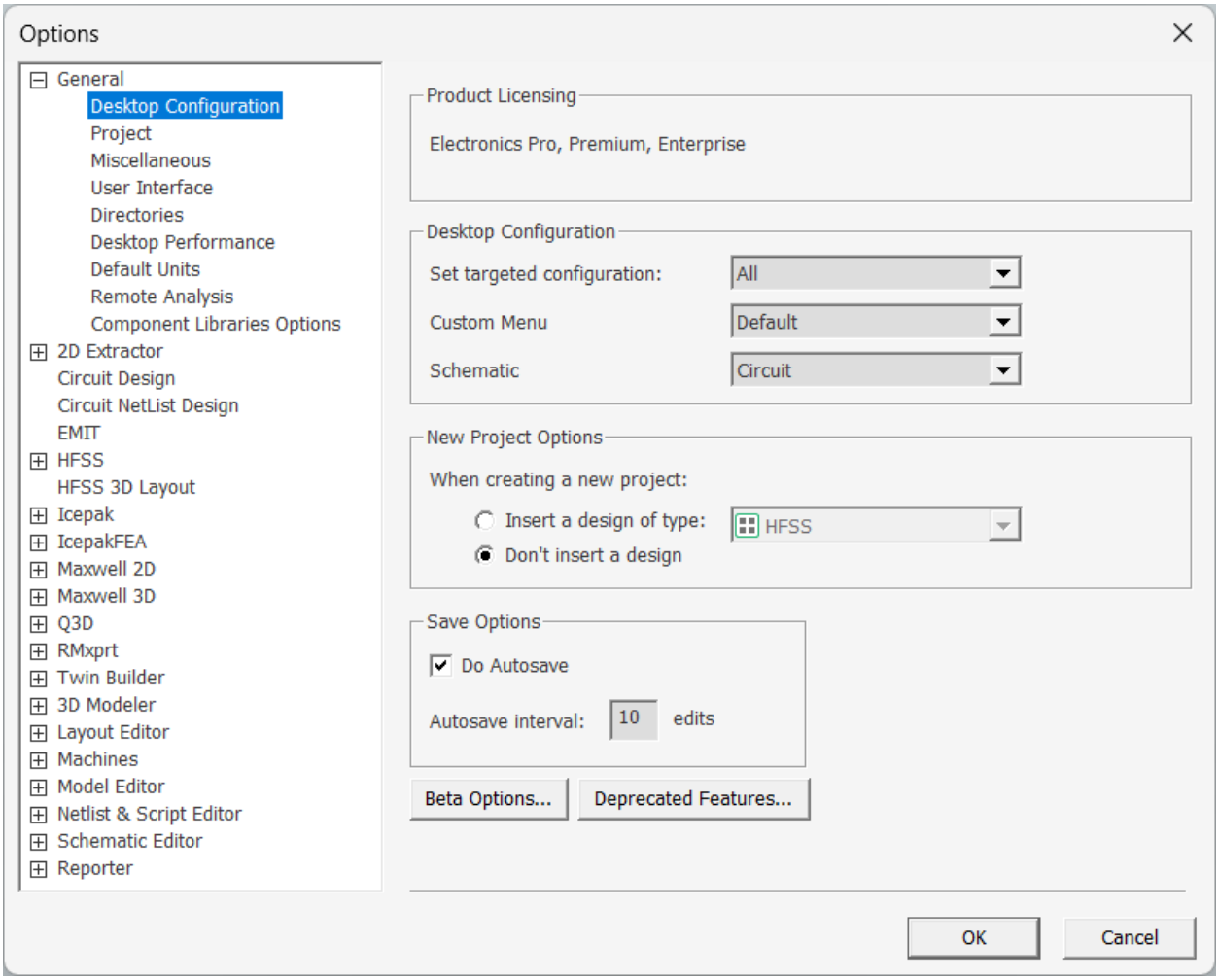
Setting General Options

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.

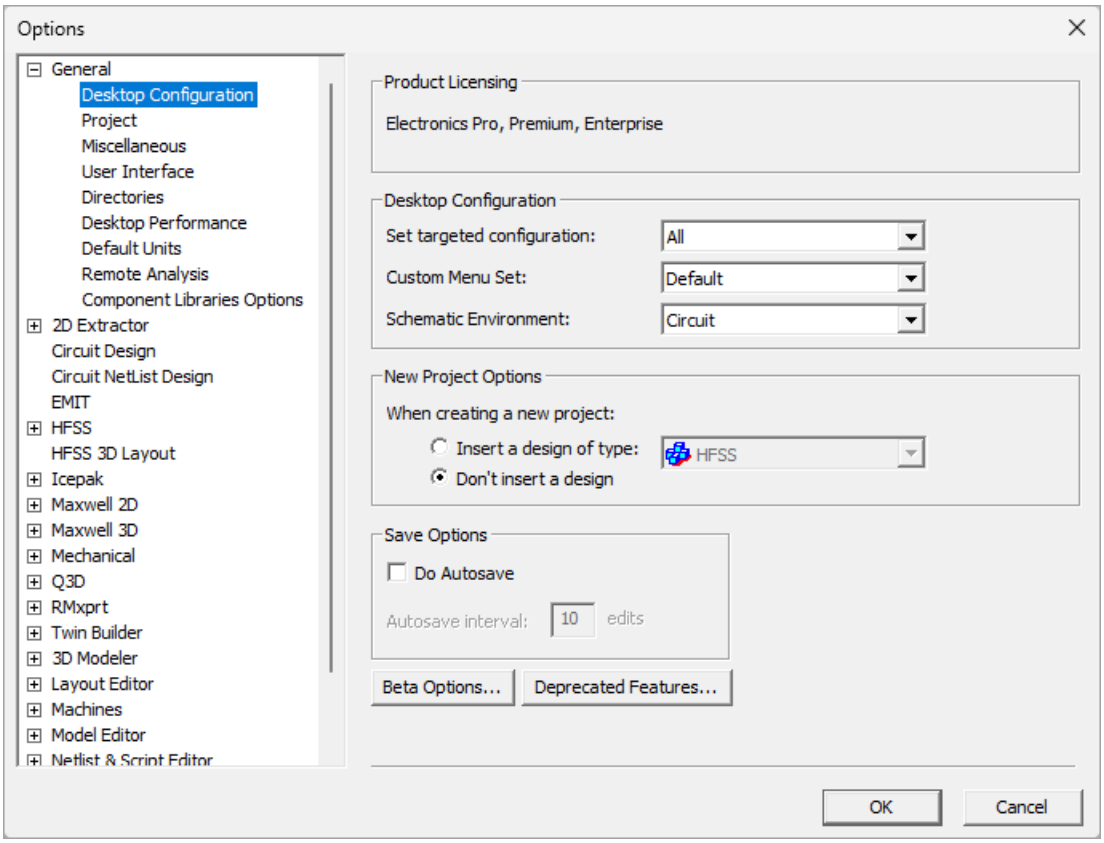
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

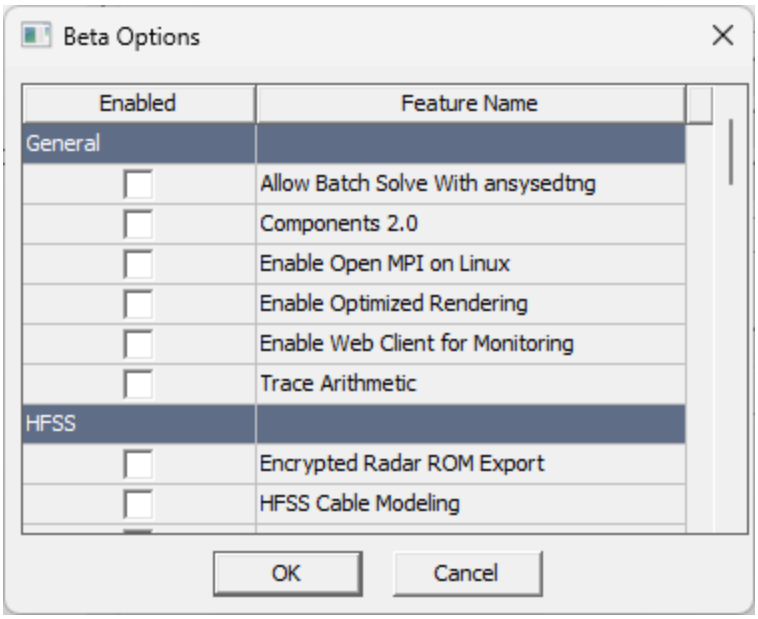
- **When Creating a New Project** – Select whether to insert a design when creating a new project. If you choose to insert a design by default, use the drop-down menu to select the default design type.

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.



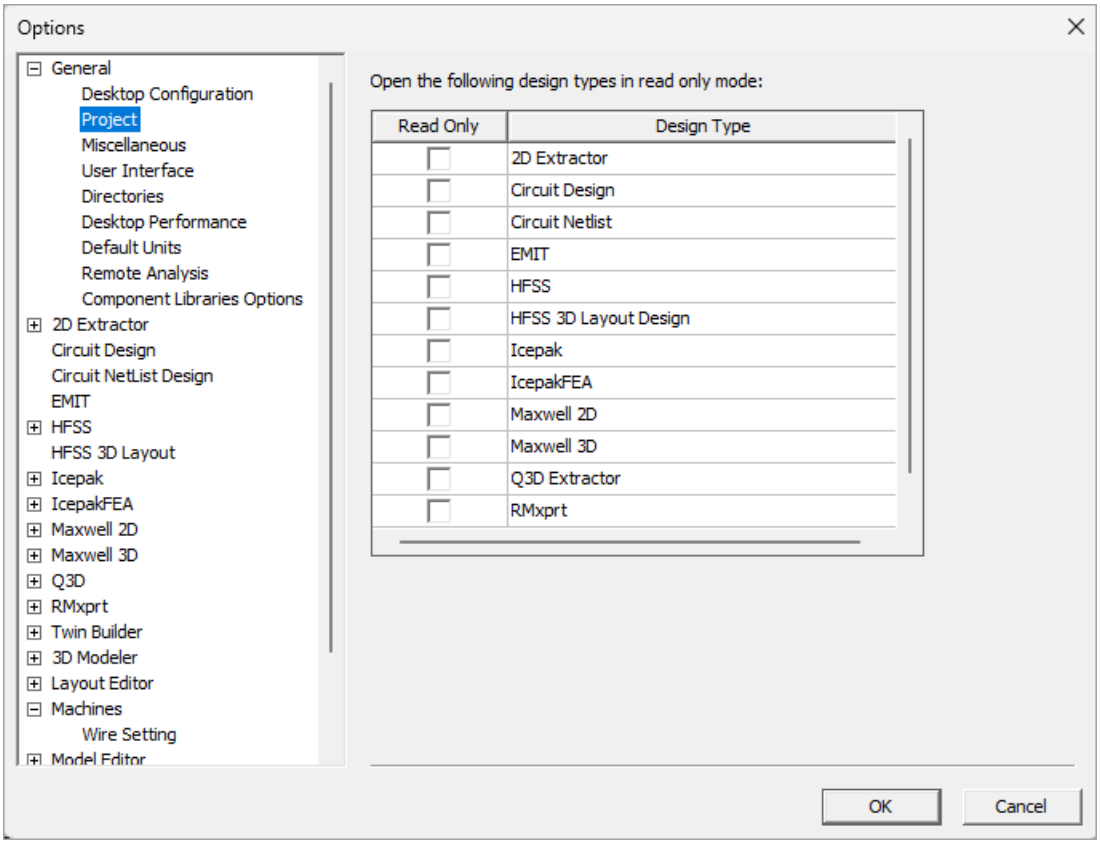
From there, enable or disable options, and click **OK**. You will need to restart to enable your selections.

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.

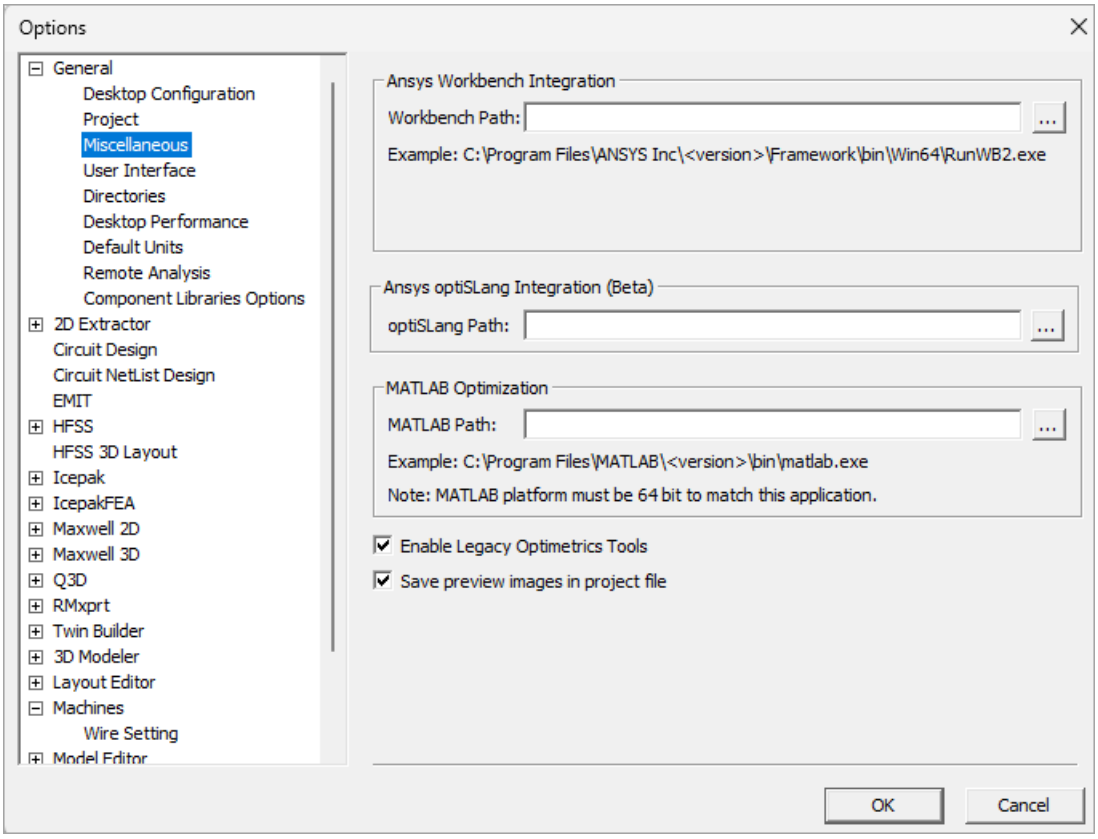
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

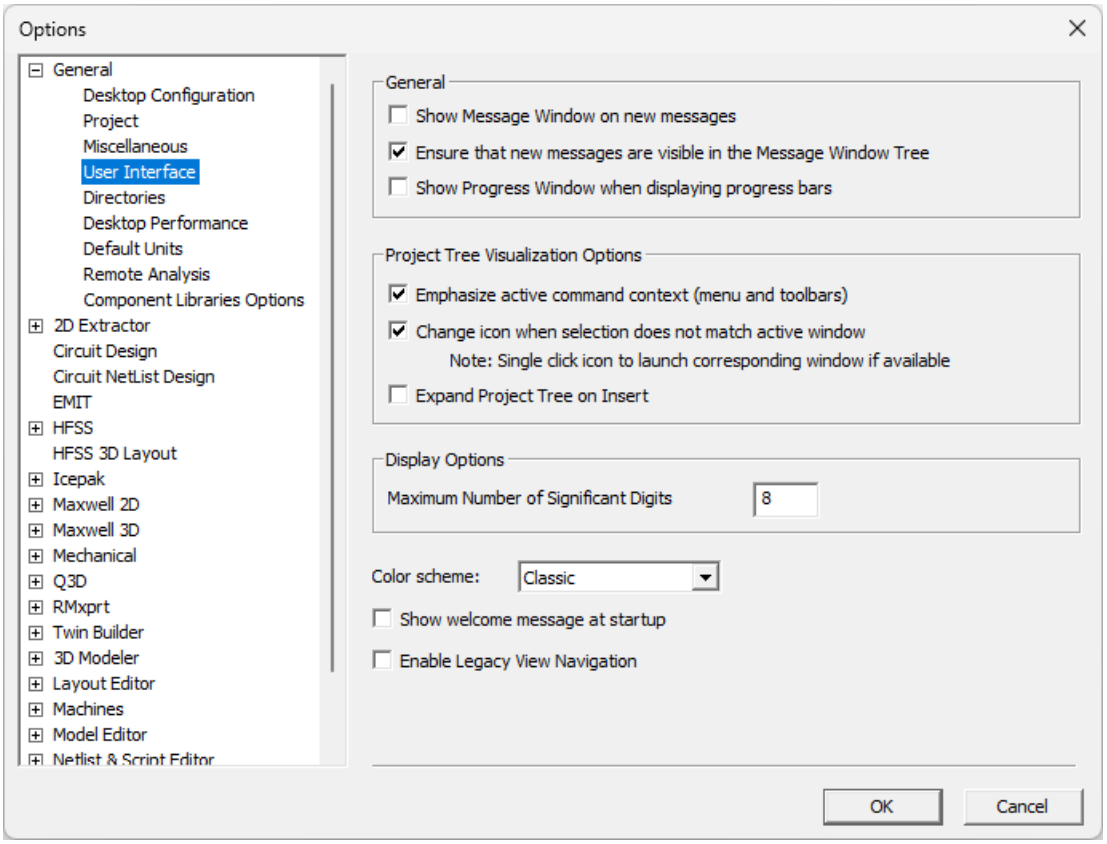
MATLAB Optimization allows you to specify the path to your installation of MATLAB. MATLAB can be used as an Optimizer.

> **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:

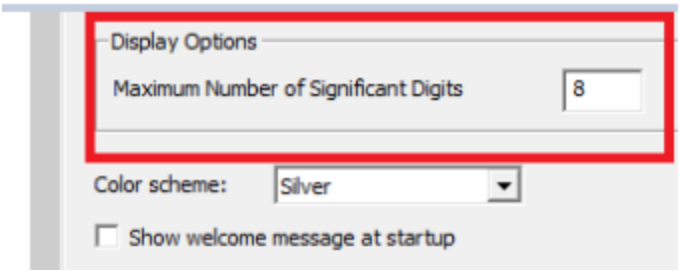
- **Show Message Window on new messages** – when selected, the message window automatically opens if a message arrives.
- **Ensure that new messages are visible in the Message Window Tree** – when selected, the message window expands as needed to display messages.
- **Show Progress Window when displaying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

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

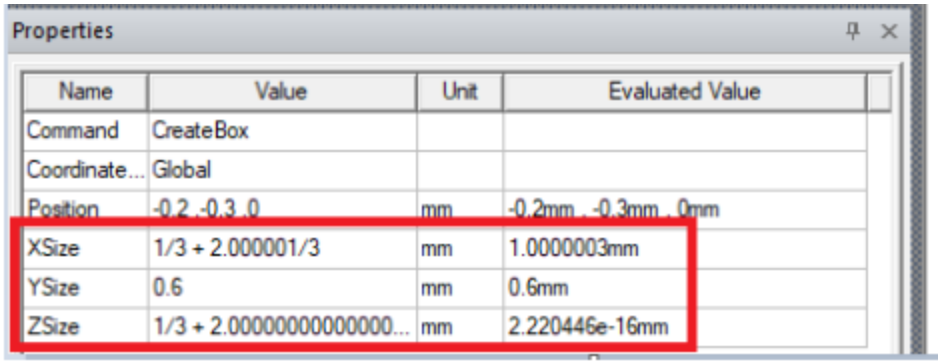
- **Emphasize active command context** – when selected, active elements in the Project Tree display in bold text.
- **Change icon when selection does not match active window** – when selected, a small, window-shaped overlay icon displays in the corner of the selected Project Tree element. This icon changes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

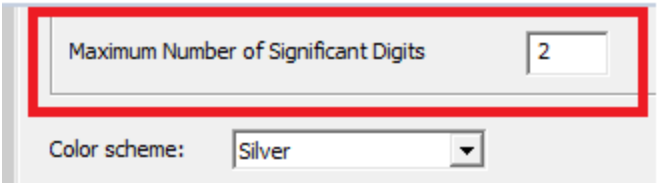
In the **Display Options** area, specify the **Maximum Number of Significant Digits** to display. The default is 8 and the maximum is 20. This affects the digits displayed in the **Solutions** dialog box, evaluated variable values, Animation dialog boxes, Optimetrics, Reports, and so forth. For example,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



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

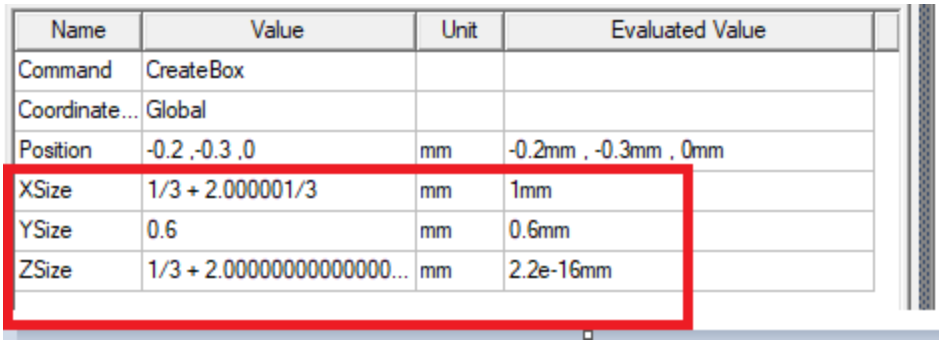


Change Precision = 2. This lets you display near-zero values in the Properties window as 0.



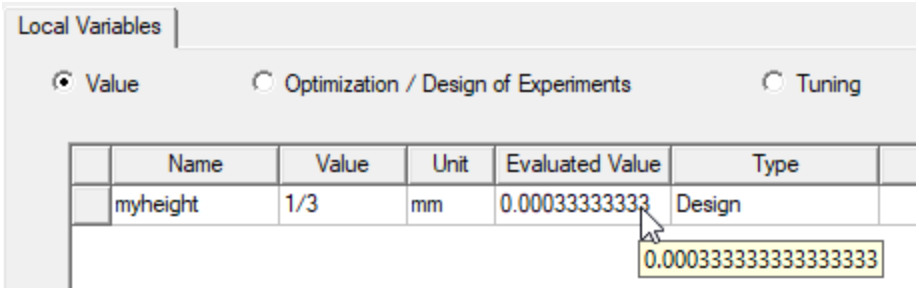
You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

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

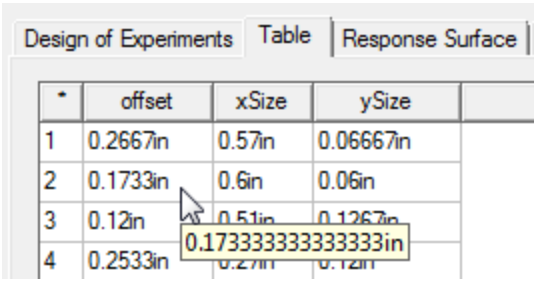


Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-0.2 ,-0.3 ,0	mm	-0.2mm , -0.3mm , 0mm
XSize	1/3 + 2.000001/3	mm	1mm
YSize	0.6	mm	0.6mm
ZSize	1/3 + 2.00000000000000...	mm	2.2e-16mm

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



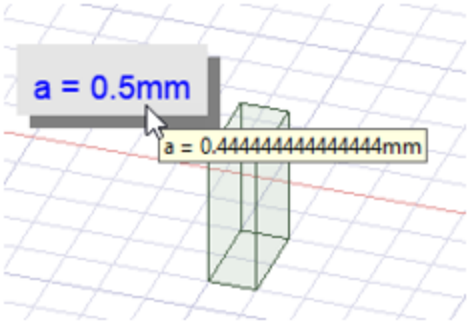
In the case of variable values, if you have assigned more significant digits, you will see these when editing the variable value. In the case of table displays of values, the tooltip display shows all available digits when the mouse pointer is over a result:

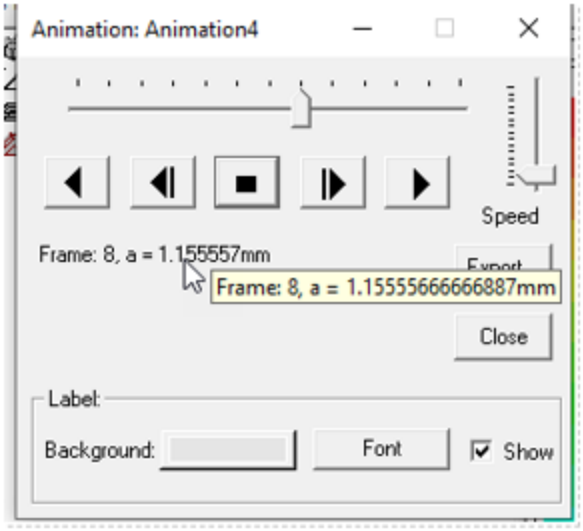


If you set the **Maximum Number of Significant Digits** to a lower value, Change Precision = 2. You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

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.

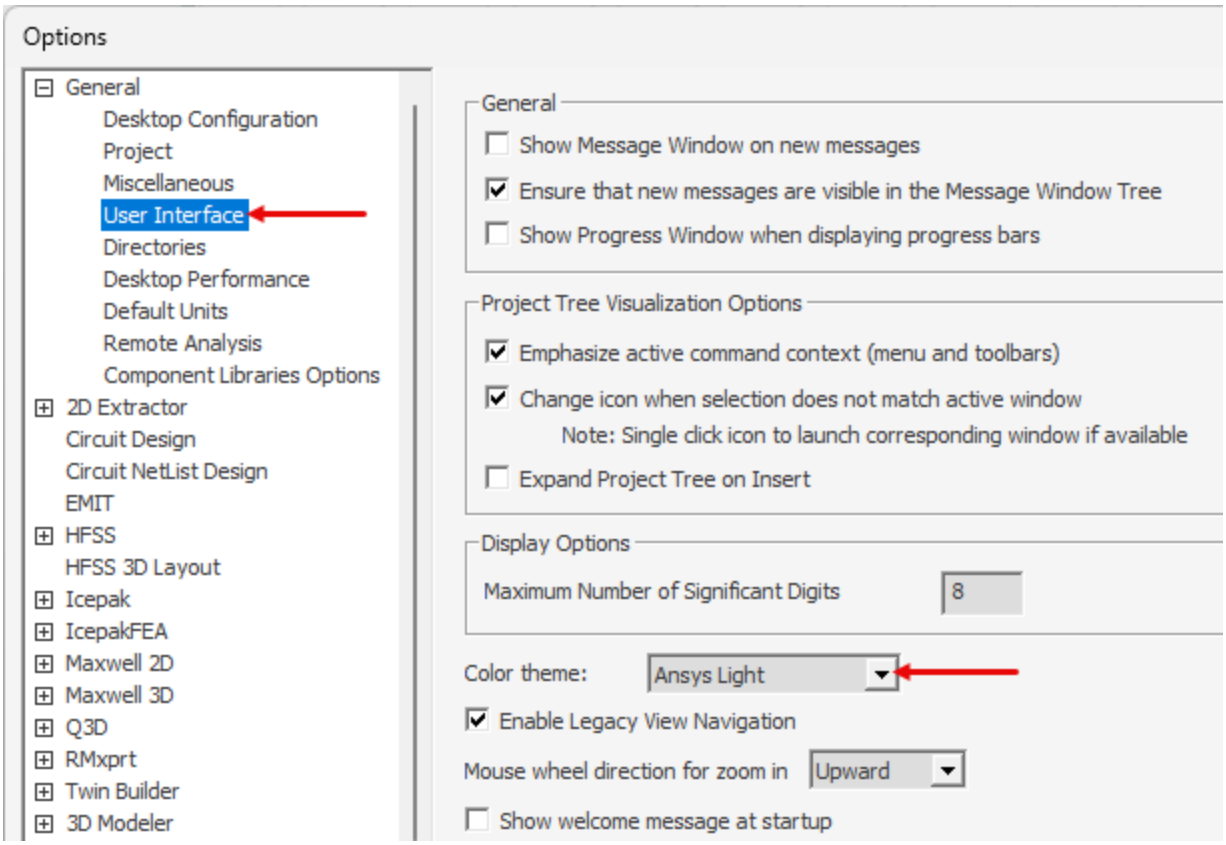
Choosing a Color Scheme

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

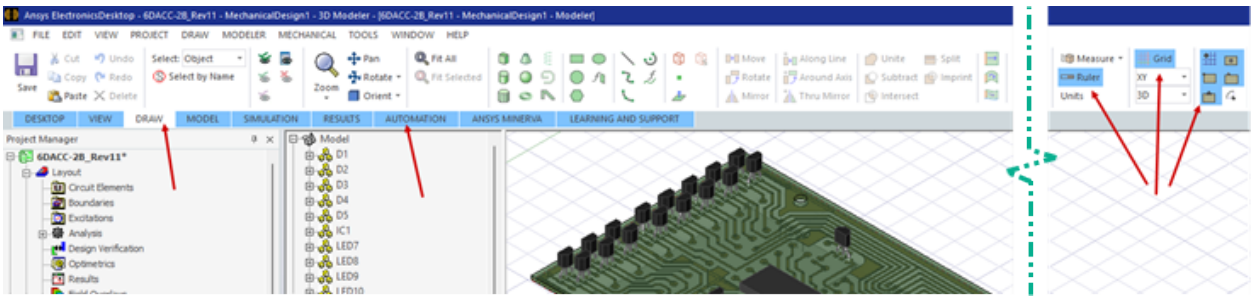
1. Access the **Options** dialog box using one of the following two methods:
   - On the **Desktop** ribbon tab, click **General Options**.
   - From the menu bar, click **Tools** > **Options** > **General Options**.

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**:

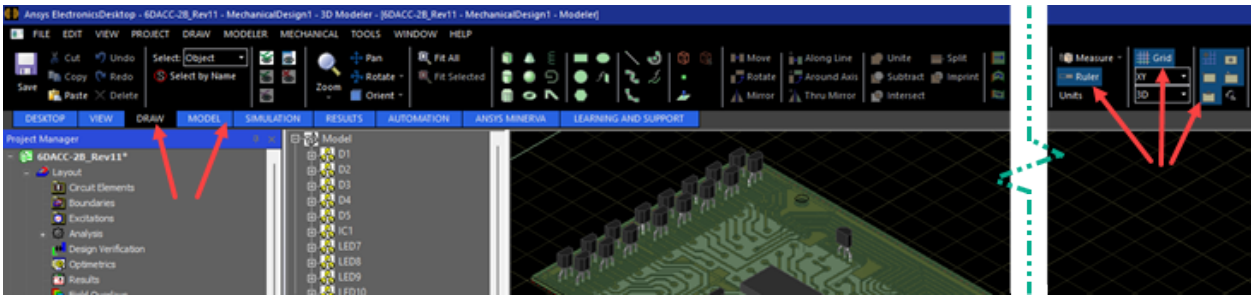


3. From the **Color scheme** drop-down menu, choose one of the following three options:
   - **Ansys Light:** A light gray and white scheme with light blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a white background (including the tab):

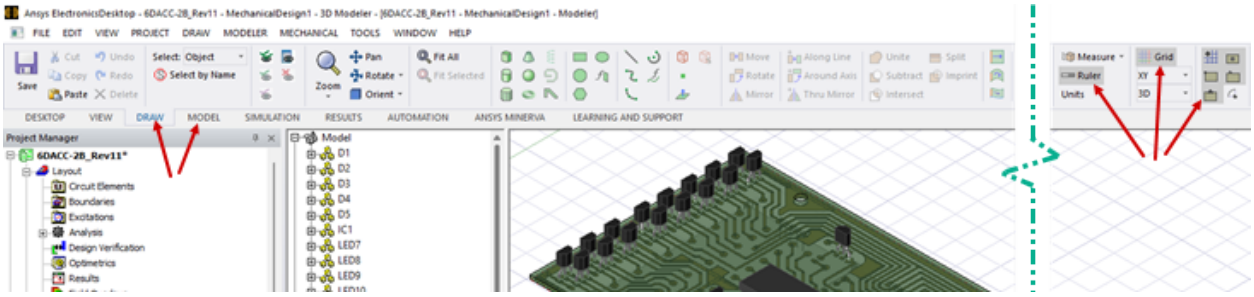


   - **Ansys Dark (beta):** A dark gray and black scheme with medium blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a black background (including the tab):

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:

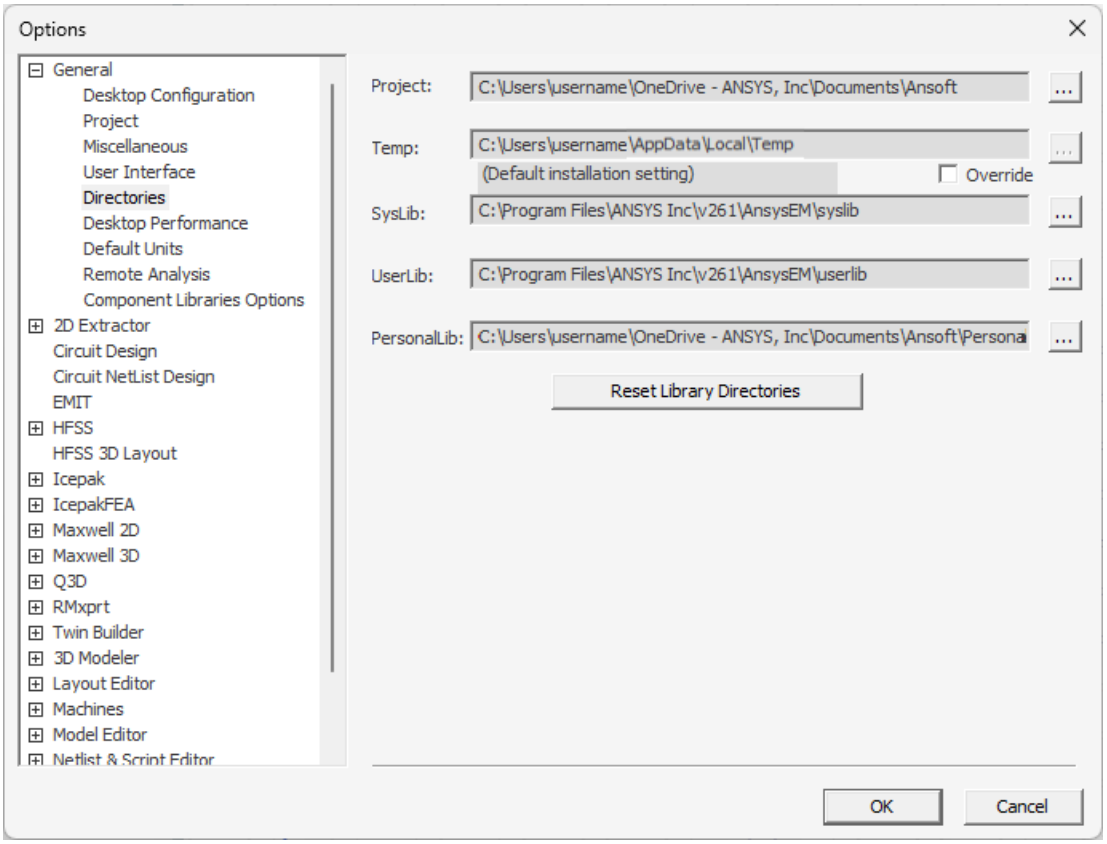


4. Click **OK** to close the **Options** dialog box.

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.

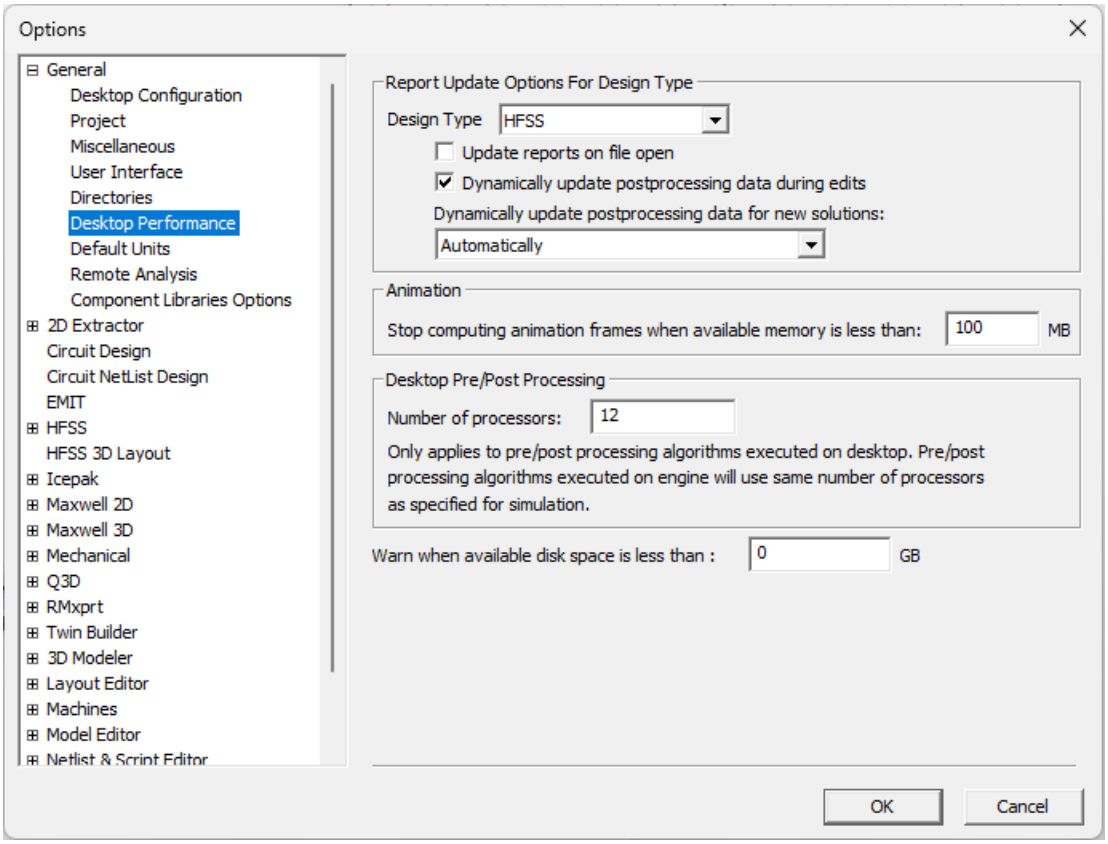
If you need to restore defaults, click **Reset Library Directories**.

> **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

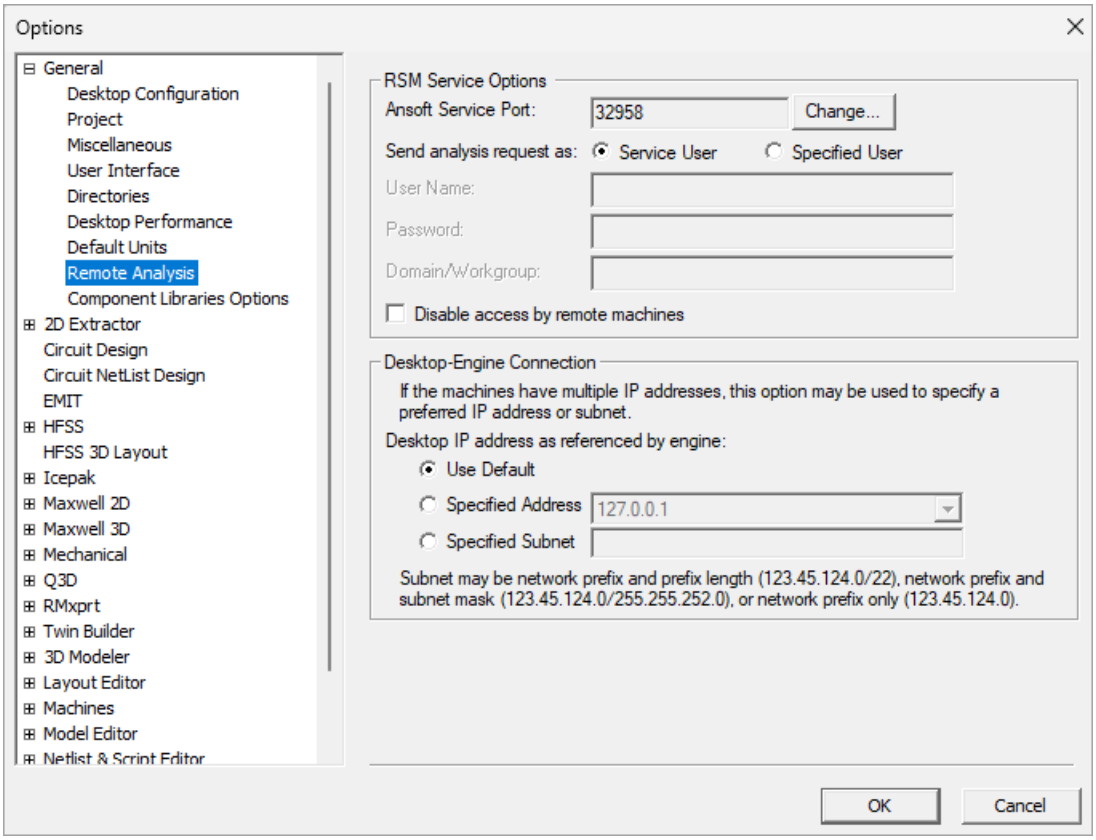
Under General Options, **Default Units** options allow you to set default units for the following:

- 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:

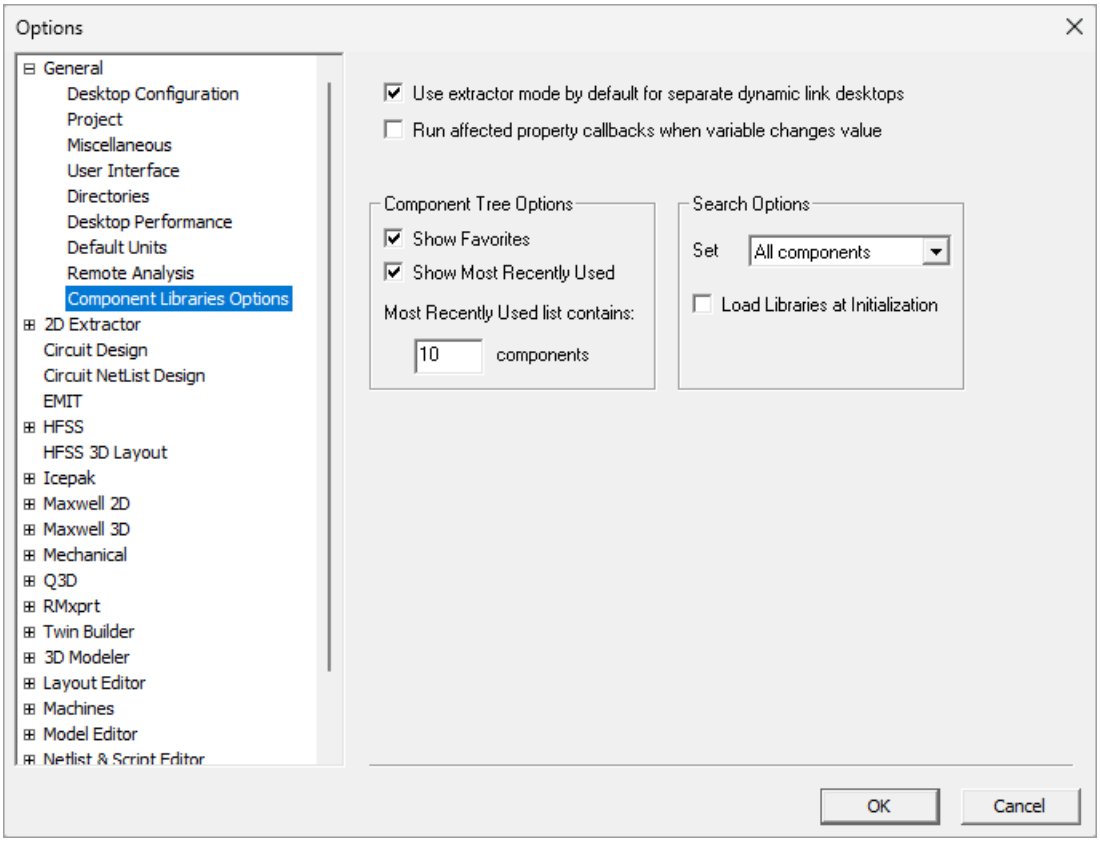
```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

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.

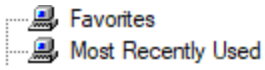
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

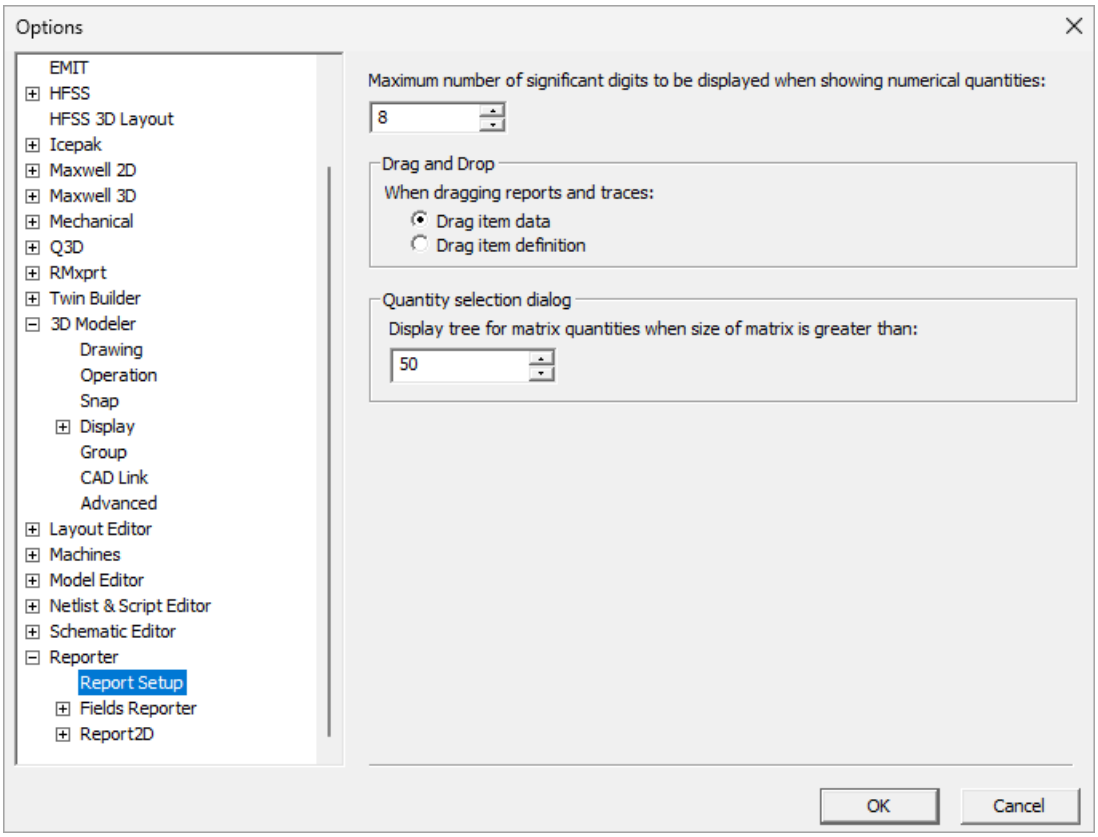
Setting Reporter Options

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.

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.



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 **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:

- Animation Options
- Mesh Plot Options
- Streamline Plot Options

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

From the **Animation** options, you can set the following:

- **Group Field Overlays by Type** – If selected, field overlays are grouped in the Project Tree. If deselected, they are not.
- **Default Phase Animation Settings** – Select the default start and stop angles as well as the number of steps for Scalar and Vector plots.

From the **Mesh Plot** options, you can set the **Clipping of volume mesh plot**. When dragging a clip plane, the plot can update automatically. Select **Never** to disable this feature, **Always** to enable it at all times, or set a maximum number of mesh elements for automatic update. This option allows you to update plots for smaller meshes but avoid automatically updating larger plots that may consume too much memory.

From the **Streamline Plot** options, set **Streamline drawing stopping criteria** and **Streamline marker spacing**.

Reporter Options: Report2D Options

Under Reporter Options, **Report2D** options fall into several categories:

- Curve
- Axis
- Grid
- Header

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

- Note
- Legend
- Marker
- Marker Table
- X/Y Markers
- Stacked
- Digital
- General
- Table

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

Report2D Options: Curve

Curve settings change how curves are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Curve**:

- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Width** – set the line width by editing the real value in the text field.
- **Arrows** – select the check box to use arrows on curve ends.
- **Show Symbol** – select the check box to have symbols mark the locations of data points on the curve.
- **Sym Freq** – set the symbol frequency by editing the integer value in the text field.
- **Sym Style** – use the drop-down menu to select a symbol from: Box, Circle, Vertical Ellipse, Horizontal Ellipse, Vertical Up Triangle, Vertical Down Triangle, Horizontal Left Triangle, and Horizontal Right Triangle.
- **Fill Sym** – select the check box to set the symbol display as solid. Otherwise it will display as hollow.
- **Sym Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Report2D Options: Axis

Axis settings change how axes are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Axis**:

- **Axis Name** – this describes the axis to which a row's settings apply. You cannot change this field.

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

- **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.

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**:

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** – 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**.
- **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**.

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 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**.
- **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

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

- **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**.
- **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

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 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.

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**.

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

- **Highlight Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Accumulate Depth** – enter a value; the default is 4.
- **Enable Y Axis Stripes** – select the check box to add stripes to the Y axis; deselect to remove stripes.
- **Auto Scale Fonts** -on by default and when enabled scales text in plots and colorkey (contour plot, field plots in 3D modeler) for high resolution screens.

Curve Tooltip

- **Show Trace Name** – select the check box to display trace names in the tooltip when hovering the cursor over a curve.
- **Show Variation Key** – select the check box to display the variation key in the tooltip when hovering the cursor over a curve.
- **Show Solution Name** – select the check box to display the solution name in the tooltip when hovering the cursor over a curve.

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.

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

- **Border Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Grid Width** – enter a value for the grid width.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Page Link Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Arrow Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Header Row

- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **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.

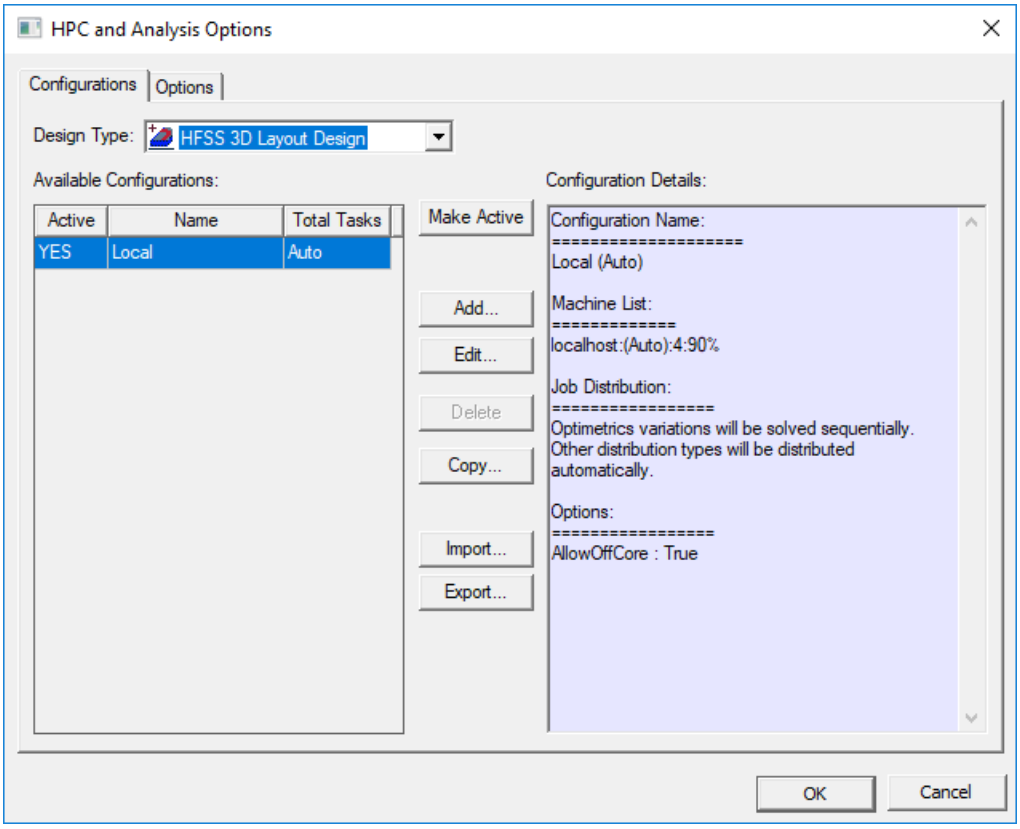
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set HPC and Analysis Options:

1. Click **Tools** > **Options** > **HPC and Analysis Options**.

   You can also access HPC and Analysis Options using the **HPC Options** icon on the **Simulation** ribbon.

   The **HPC and Analysis Options** window appears, displaying the **Configurations** tab.



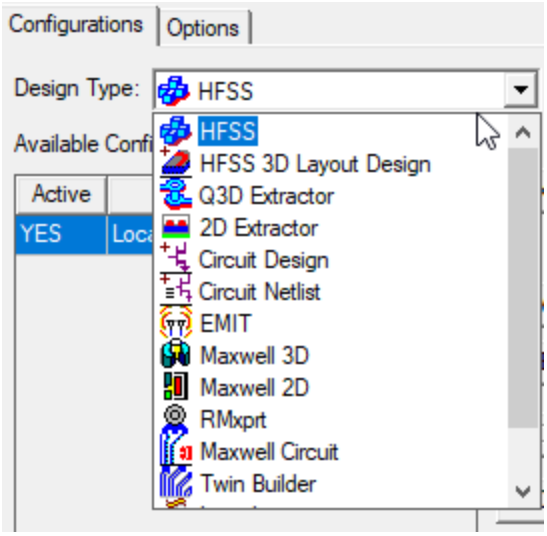
Configurations Tab

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.

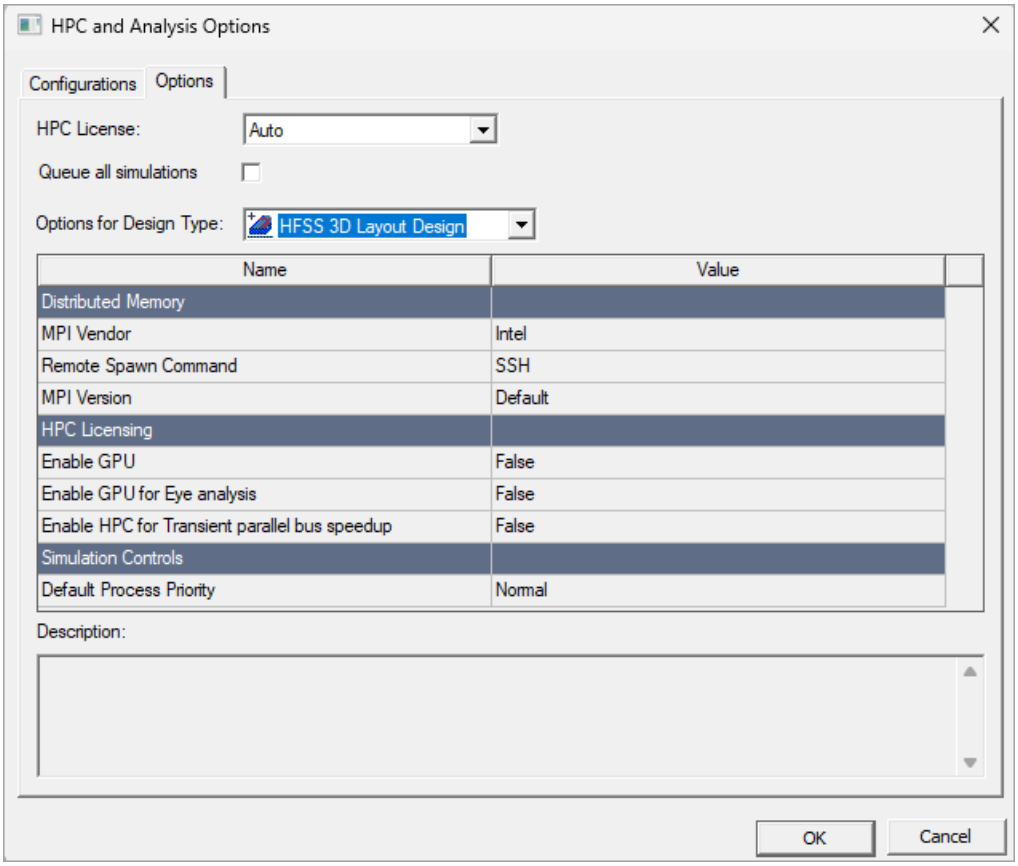
> **Note:**
>
> Importing analysis configurations always adds the imported analysis configurations to the current design type. If there is a name conflict between an imported analysis configuration and an existing analysis configuration, the imported configuration is renamed and a notification appears.

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:

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

- A design of the matching design type is being solved.
- You have not specified corresponding overriding batch options on the command line.

From the **Options** tab, you can:

- Choose the HPC license type
- Enable queuing
- Specify Distributed Memory settings (for example, MPI for certain solvers)
- Set licensing options
- Enable GPU (for Transient, Matrix and SBR+ solves)
- Set the Default Process Priority

> **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.

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

HPC licenses enable all parallel and distributed simulations, including distributed variations. Distributed variations require a single set of solver licenses, plus HPC to enable the variations.

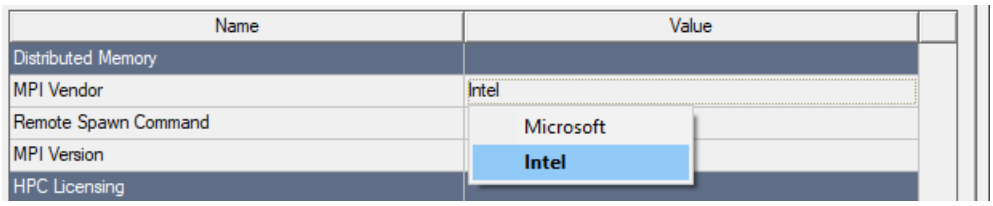
For HPC Workgroup, distributing *N* variations requires 8*(*N*–1) workgroup licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* workgroup licenses will enable one additional HPC unit per variation. For HPC Pack, distributing *N* variations requires *N*–1 pack licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* pack licenses will enable 8, 32, 128,... additional HPC units per variation.

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

> **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:



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.

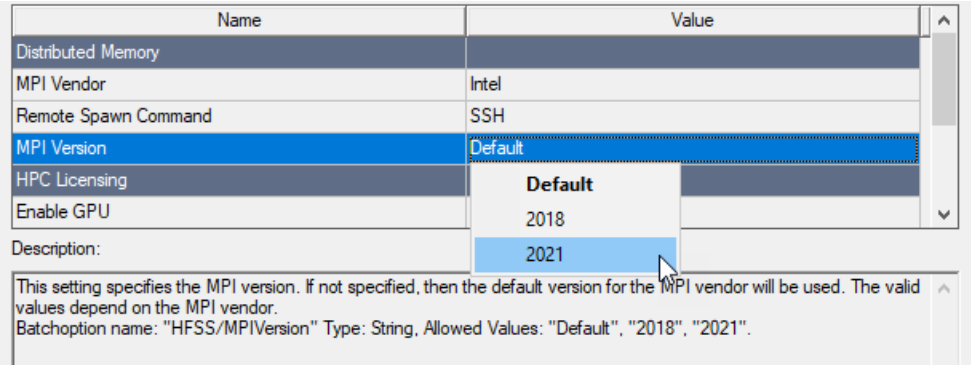
To use the distributed memory solution you will need to install MPI software from a supported third party vendor on all the machines you intend to use.

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 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.

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

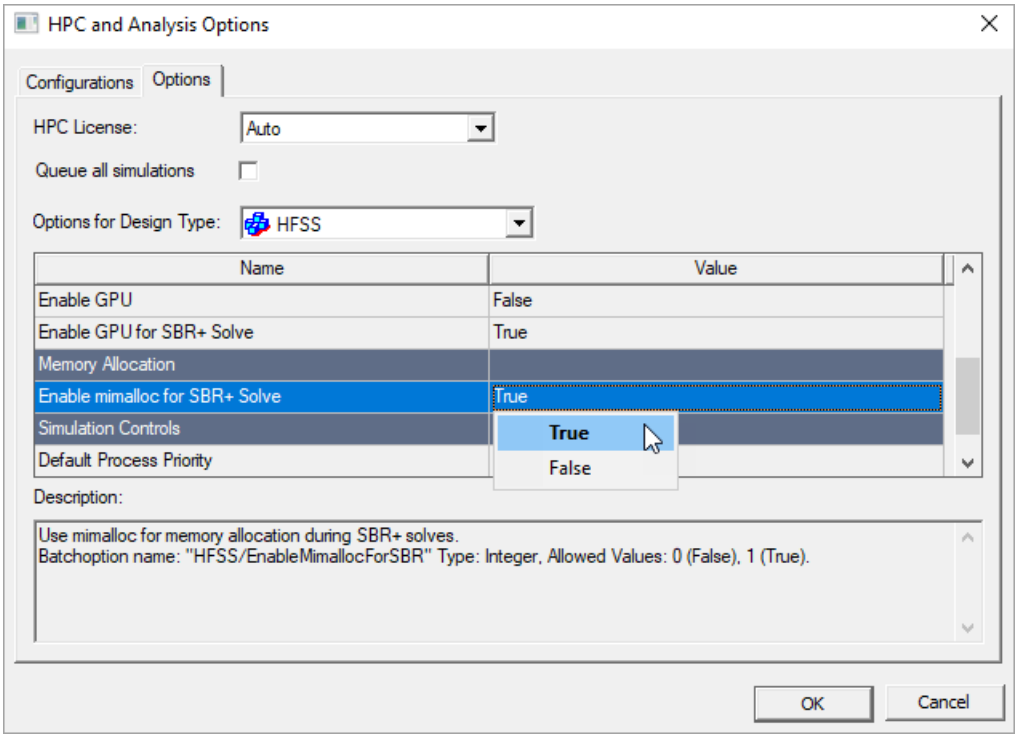
By default, MPI vendors use the fastest interconnect by default (typically InfiniBand is faster than Ethernet). If you want to override the default behavior and force the use of Ethernet, you can set the ANSOFT_MPI_INTERCONNECT environment variable to "eth" for the job. Also see Setting Intel MPI Interconnect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

1. 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

2. **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.



3. **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.

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, HFSS 3D Layout 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 HFSS 3D Layout 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.

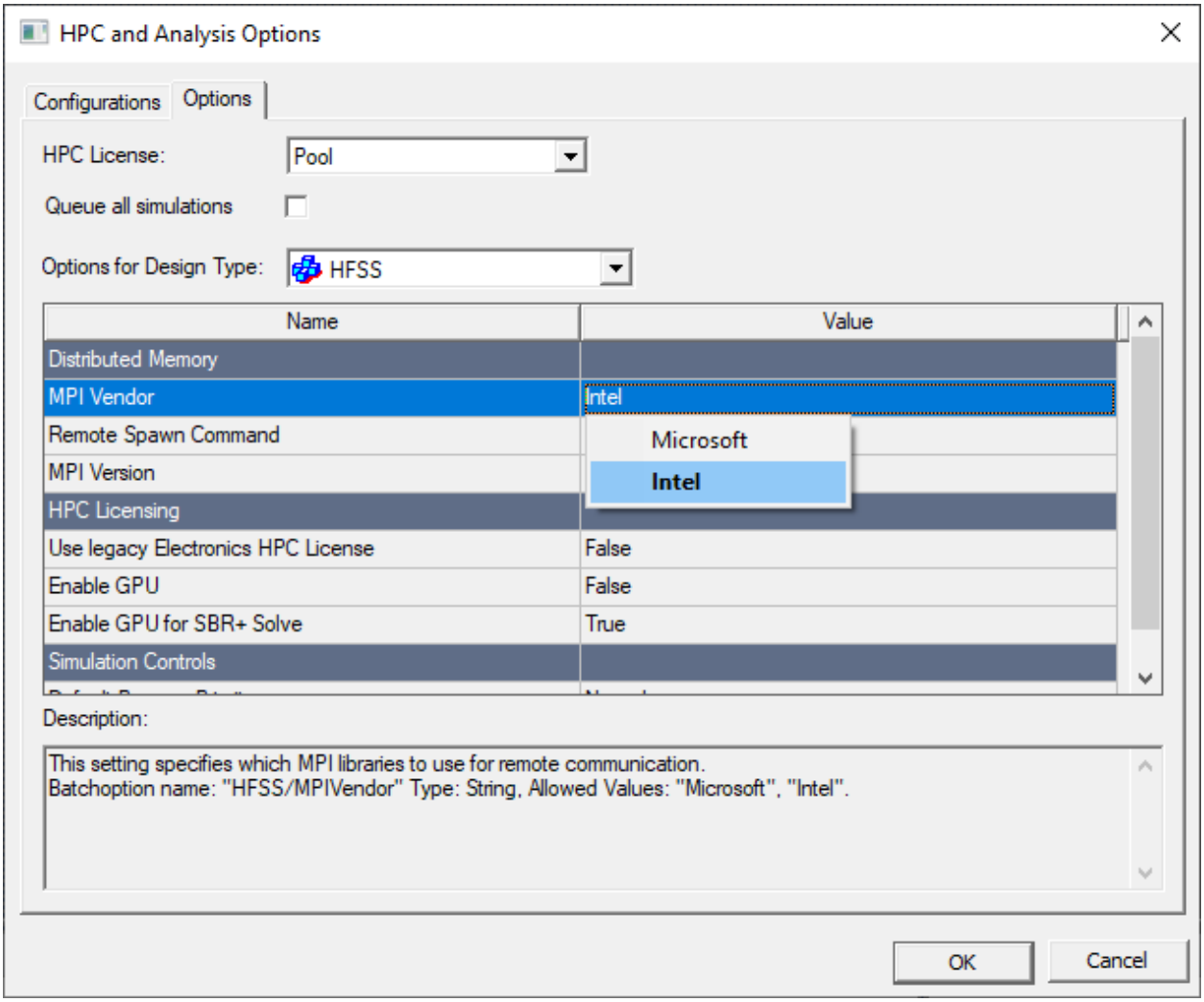
MPI Vendor and MPI Version

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.

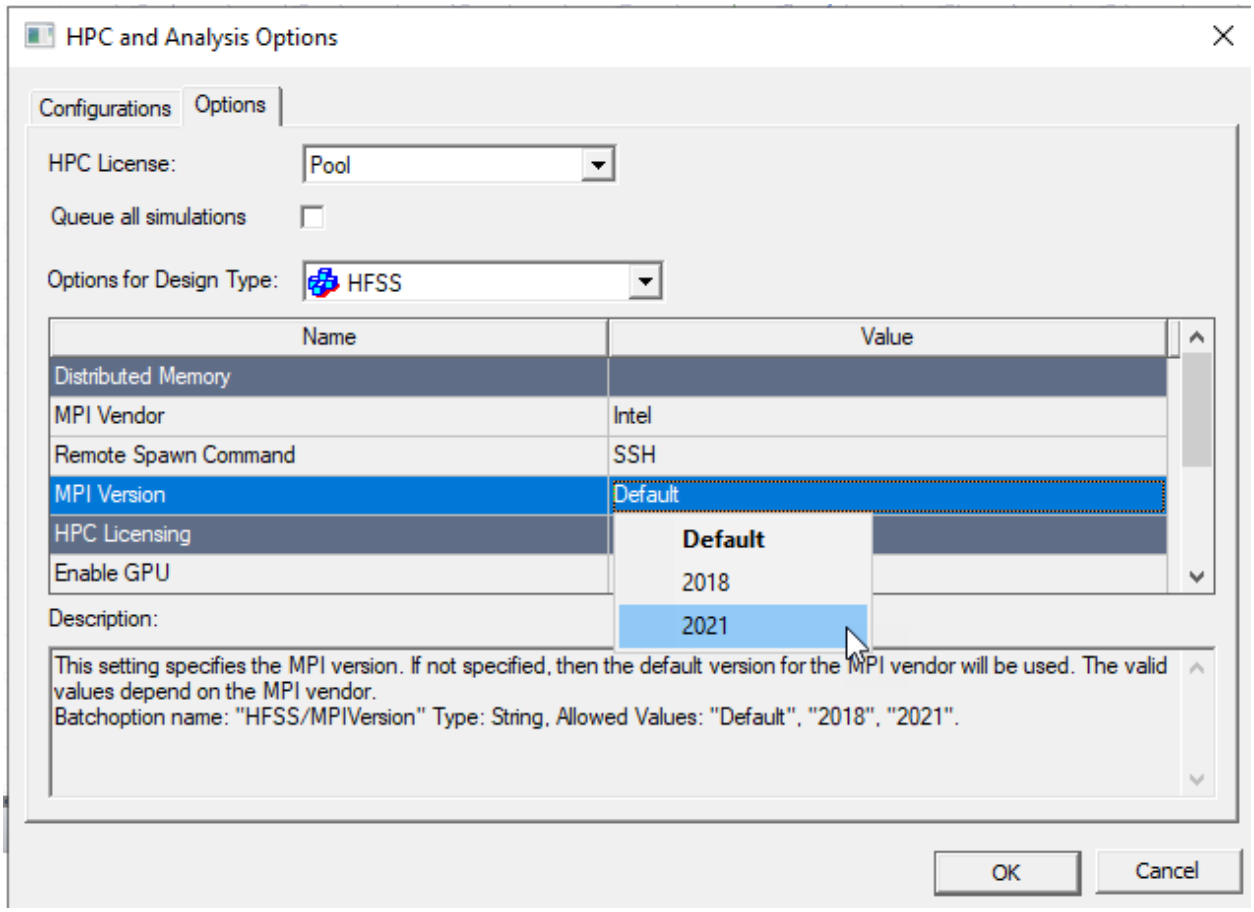
MPI Version

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.

Setting Options via Configuration Files

In addition to setting options from the Electronics Desktop user interface (UI), you can also set options in several configuration files. Option settings in configuration files may apply to all users or only to a specific user, and they may apply to all hosts or only to specific hosts. There are four levels of settings, listed below from most specific (highest precedence) to most general (lowest precedence):

- Host-dependent user options – apply to the specified user on the specified host only
- Host-independent user options – apply to the specified user on all hosts
- Host-dependent default options – apply to all users on the specified host
- Installation default – default for all users on all hosts

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

In the list above, settings at any level override settings at lower levels . If there is no setting in any file, the application default value is used. See UpdateRegistry for instructions on selecting these levels.

> **Important:**
>
> Options set from the Desktop UI override and update user settings in configuration files. Otherwise, the existing configuration file settings are used.

Behavior Examples

Consider running an application as user **jsmith** on host **host123**. If there is no host-dependent user setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for user **jsmith** on host **host123**, but there is a setting for the "Expand Project Tree on Insert" option in the host-independent user options config file for user **jsmith**, then the latter setting will be used if it is not overridden using the Desktop UI. Any settings in the host-dependent default options config file or the installation default config file are ignored.

As another example, consider running an application as user **jdoe** on host **host123**. If there is no setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for **jdoe** on **host123** or in the host-independent user options config file for user **jdoe** or in the host-dependent default options config file for host **host123**, then the value from the installation default config file are used, if present.

Rules for Modifying Option Settings

Option settings displayed in the Desktop UI follow the above rules. That is, if there is a setting in any of the option config files, then the setting from the highest priority config file is displayed in the Desktop UI. If there is no setting in any of the option config files, then the global default value is used. You can modify settings using the Options dialog boxes in the Desktop UI. If the dialog box is closed with the **Cancel** button, changes made on any of the tabs are discarded. If the dialog box is closed with the **OK** button, any settings that have been changed by the user are written to the host-dependent user options config file. The changed values written to this file are then used the next time that the application is run by the same user on the same host. The Desktop UI option settings are not written to any of the other option config files.

Configuration File Locations

The configuration files for host-dependent and installation **default** options reside at: "<*installation-directory*>\ANSYS Inc\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.

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

For products that have multiple Desktop versions, each will have a separate user-specific config directory, with a different value for the <*ApplicationName&Version*> directory name.

Table of Directories and Files

The following table shows the directories and files, where the *Level Name* is the name used to describe an options config file when using the UpdateRegistry tool.

Config File	**Level Name**	**File Name**	**Windows Directory Path**	**Linux Directory Path**
host-dependent user options	user_machine	<*hostname*>_user.XML	%UserProfile%\Documents\Ansoft\<*ApplicationName&Version*>\config	$HOME/Ansoft/<*ApplicationName&Version*>/config
host-independent user options	user	user.XML		
host-dependent default options	install_machine	<*hostname*>.XML	*<Installation_Directory>*\ANSYS Inc\<*Version*>\AnsysEM\config	*<Installation_Directory>*/ansys_inc/<*Version*>/AnsysEM/config
installation default	install	default.XML		

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)
- ***<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)

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		

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 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.

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,

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 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.

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>
]
```


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

Required:

`<name>`	The application or product name and version. For example, 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.

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.

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

-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

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.

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

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:

- 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:

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 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 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.

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

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 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

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

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:

```
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
```


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

User Options and the UpdateRegistry Tool

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

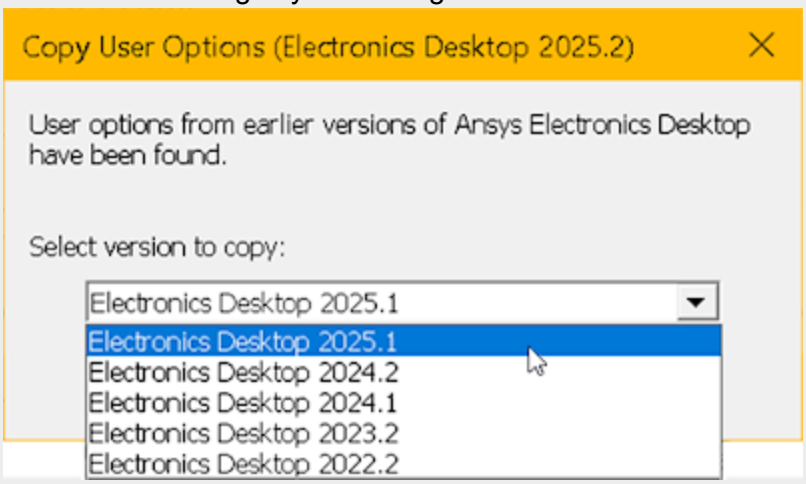
When you change an option's value using the Ansys Electronics Desktop UI, the new value is stored in the currently active user options registry. You can use the UpdateRegistry tool to add, modify, or remove settings. However, you cannot use the Desktop UI to *remove* settings from the registry. *You 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. We specify a desired RAMLimitPercent of 75 for this analysis.

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/RAMLimitPercent'=75
'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

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

4 threads for the non-distributed part of the analysis, so NumberOfProcessors=4. Because we do not use multiprocessing for the distributed analysis, we will run four distributed COM engines on each host, with a singe core available for each engine. We will specify a RAM limit of 75% for this analysis.

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/'RAMLimitPercent'=75
'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.

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 example, we run a batch analysis of project file testproject.adsn on the local host. We want all temporary files and directories created in directory C:\temp\ansysedt instead of using the installation default temporary directory. Because the local host has four cores, we decide to use four threads for multiprocessing, for both distributed and non-distributed parts of the analysis.

Here is a sample command line for this analysis, where the project file \\host123\projects\testproject.adsn is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Local -batchoptions
"TempDirectory='C:/temp/ansysedt'
'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:

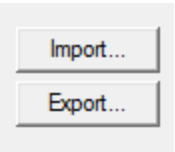
  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

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 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.

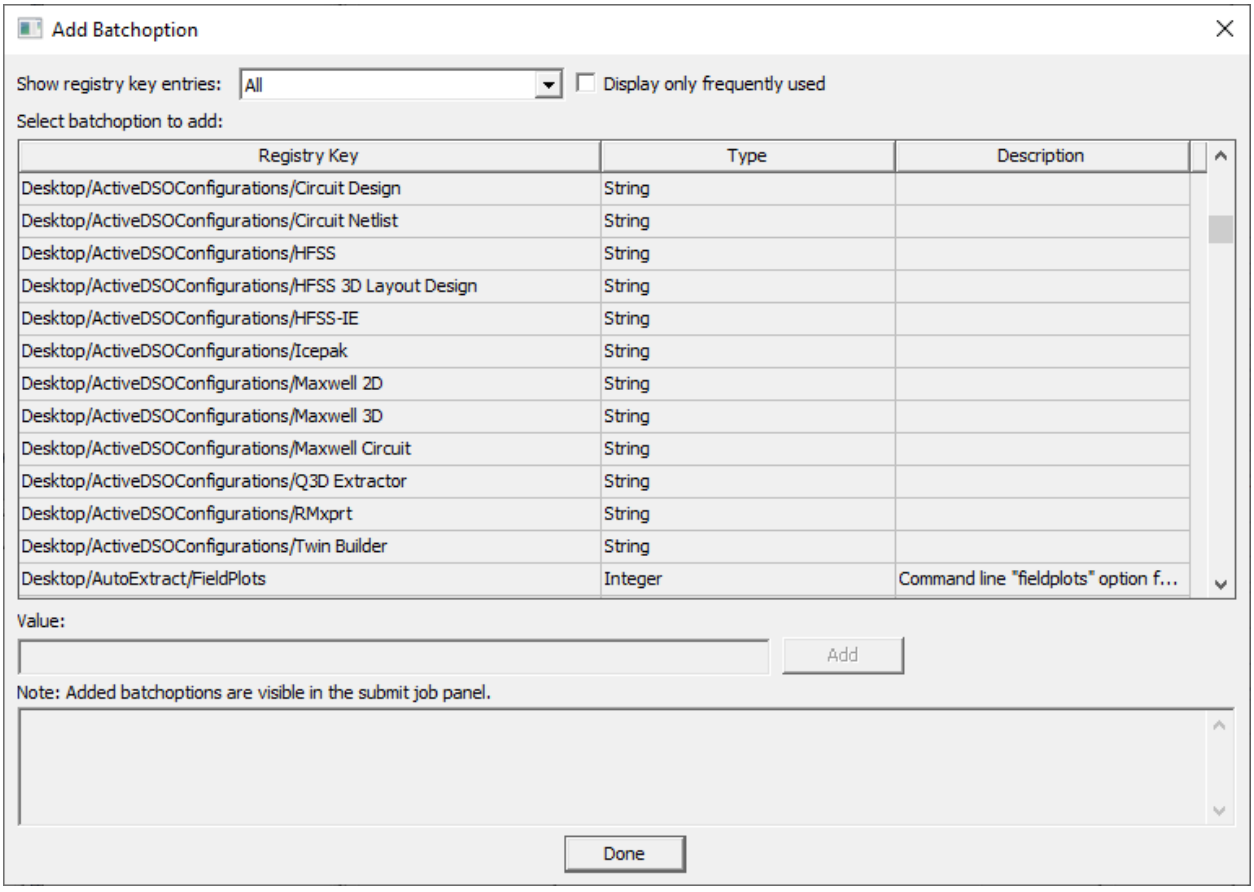
This procedure also works from the Linux <*installation_directory*>, which has the same downstream path (for example, under "opt/") as a Windows installation (typically under "C:\Program Files\").

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

Note:

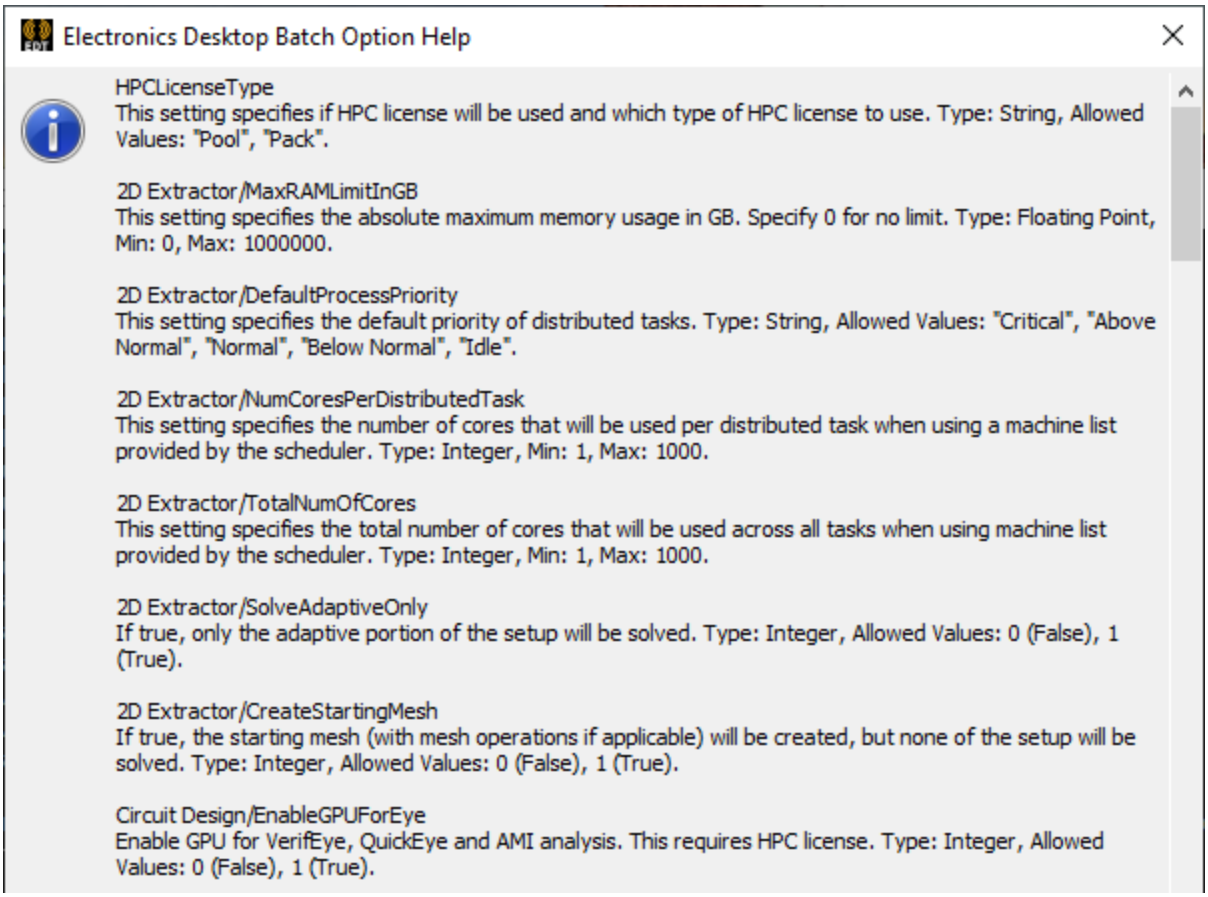
For special product-specific options that are neither available from the GUI nor listed by the UpdateRegistry tool, see Special Batch Options. Even though the -GetKeys switch does not list these special options, you can still use the UpdateRegistry tool to set them.

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

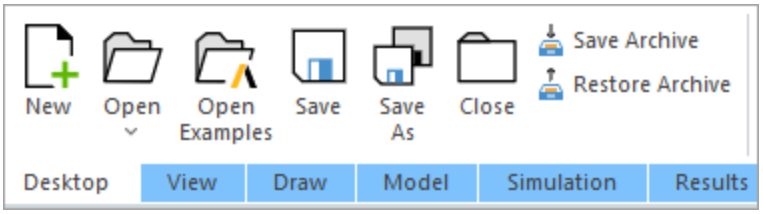
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Updating Design Components
- Undoing and Redoing Commands
- Inserting a Documentation File
- Saving Project Notes
- Printing

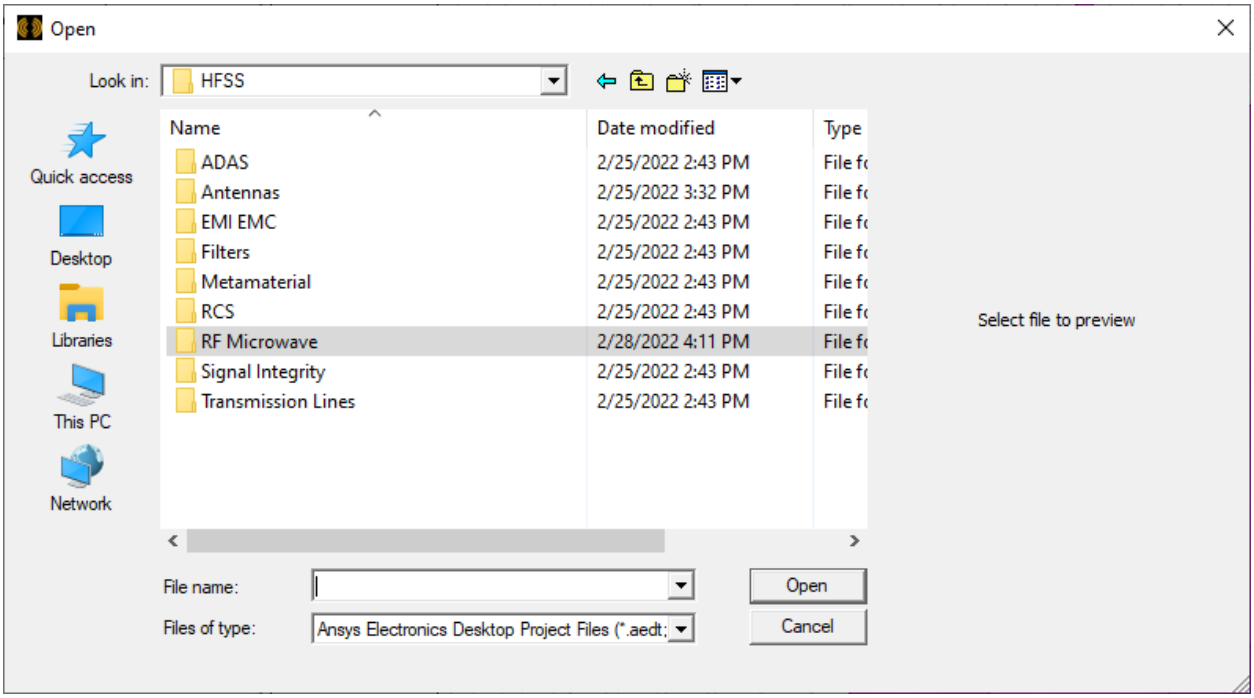
Opening Projects

Open a previously saved project using the **File** > **Open** command.

1. Click **File** > **Open** or click the **Open** icon on the **Desktop** tab of the ribbon.



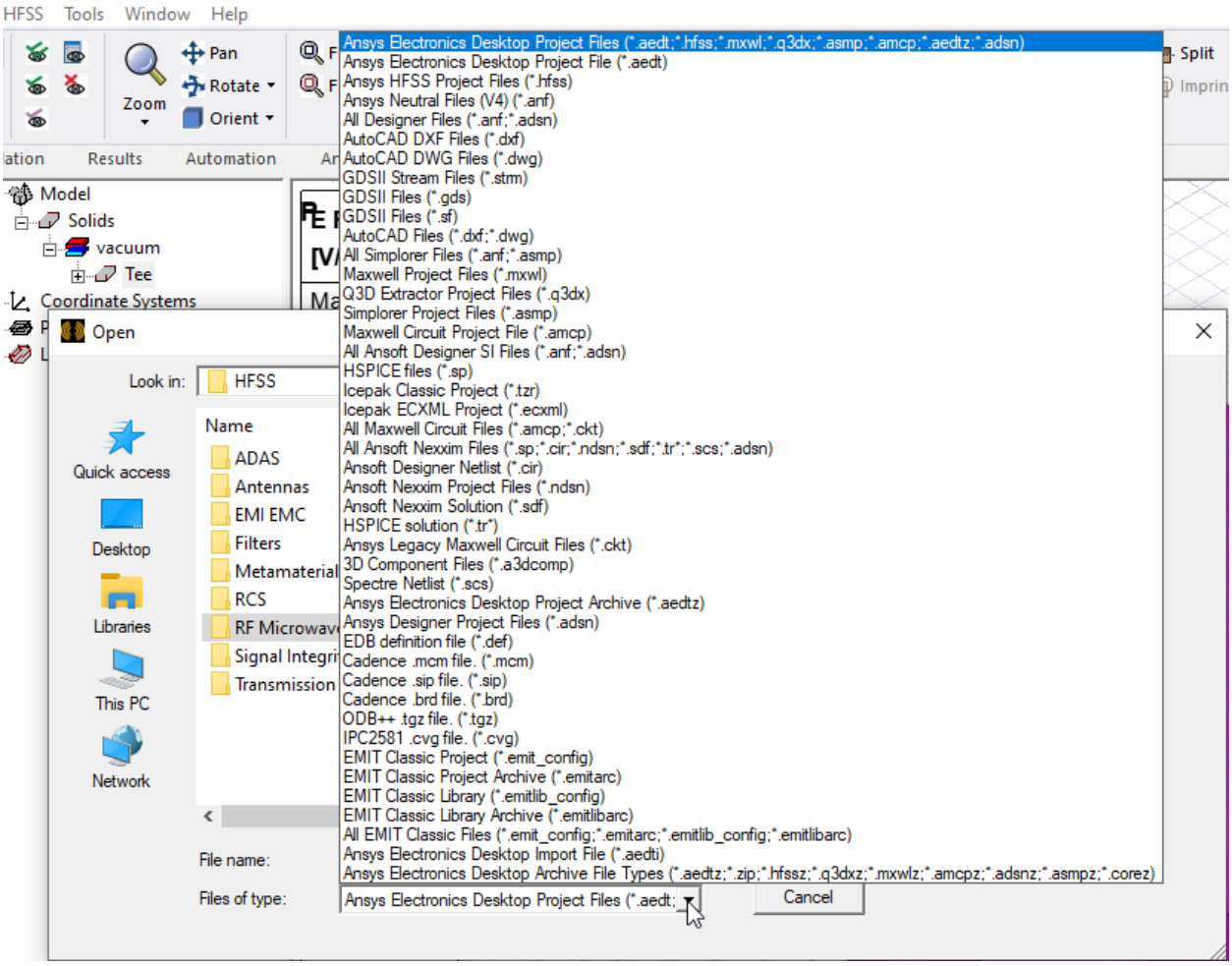
2. Use the file browser to find the project file.



By default, only files that can be opened or translated display. *.aedt is the main default, although, you can also view several types of legacy files. The drop-down menu for Files of

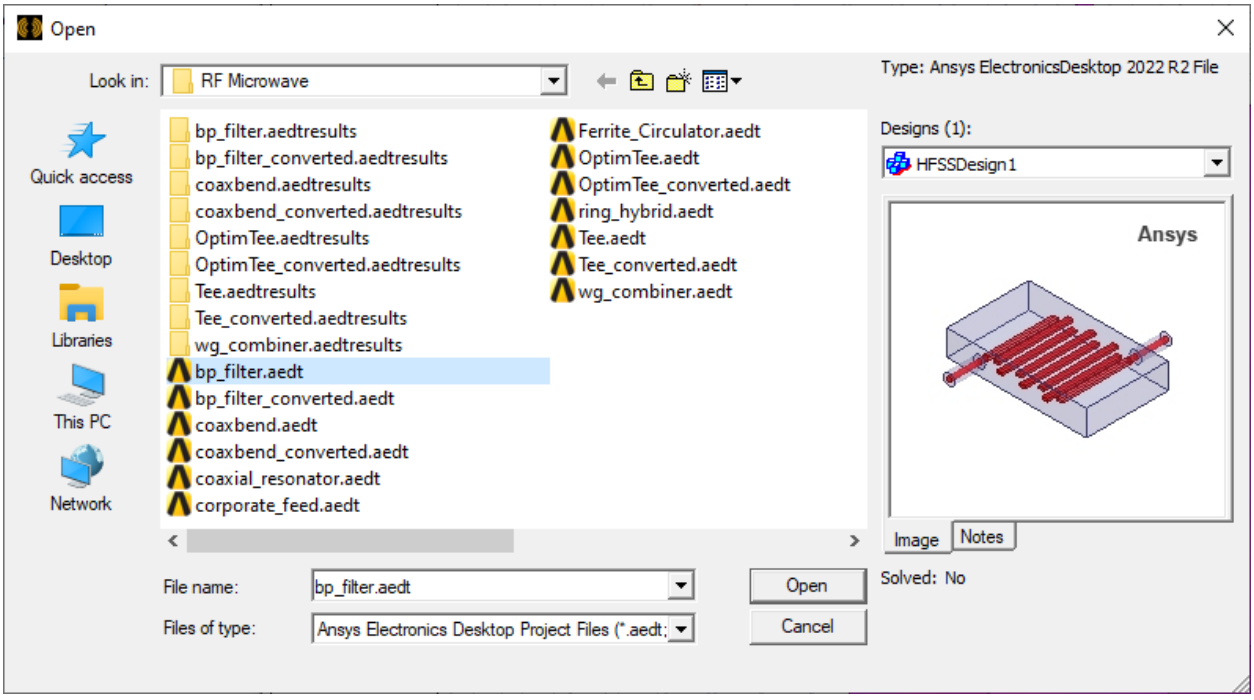
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

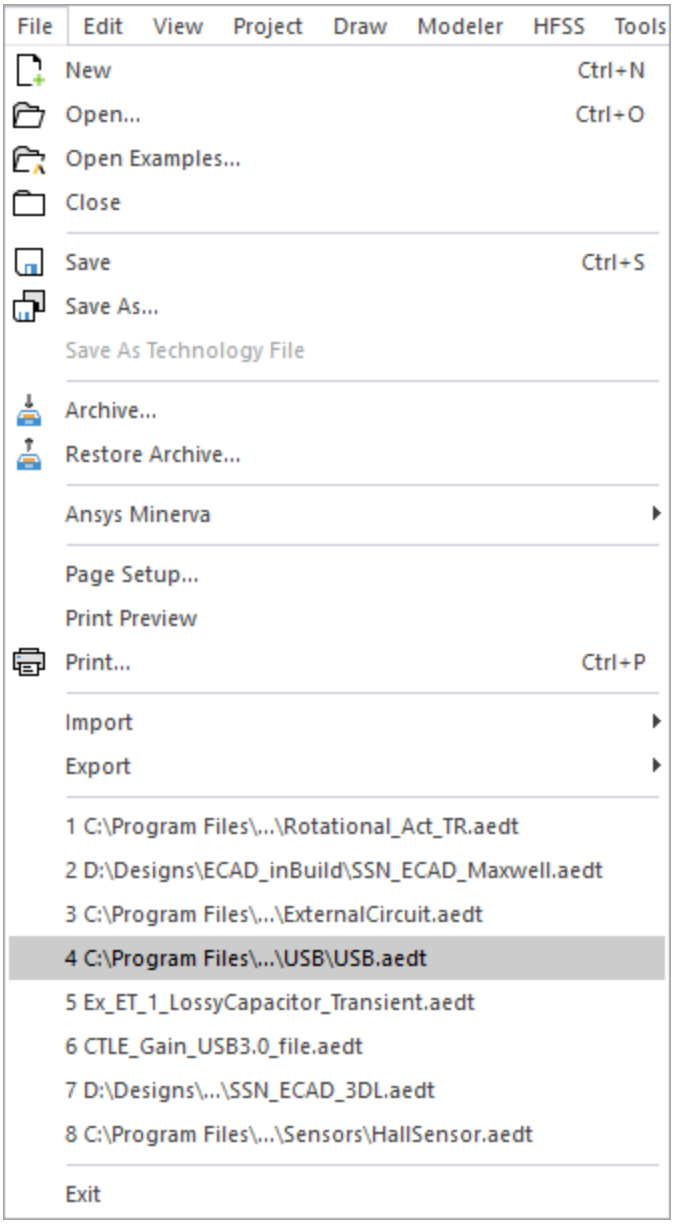
You can also open a saved project by:

- Dragging a project file icon to the Open icon
- Dragging a project file icon to the desktop
- Double-clicking a project file icon

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.



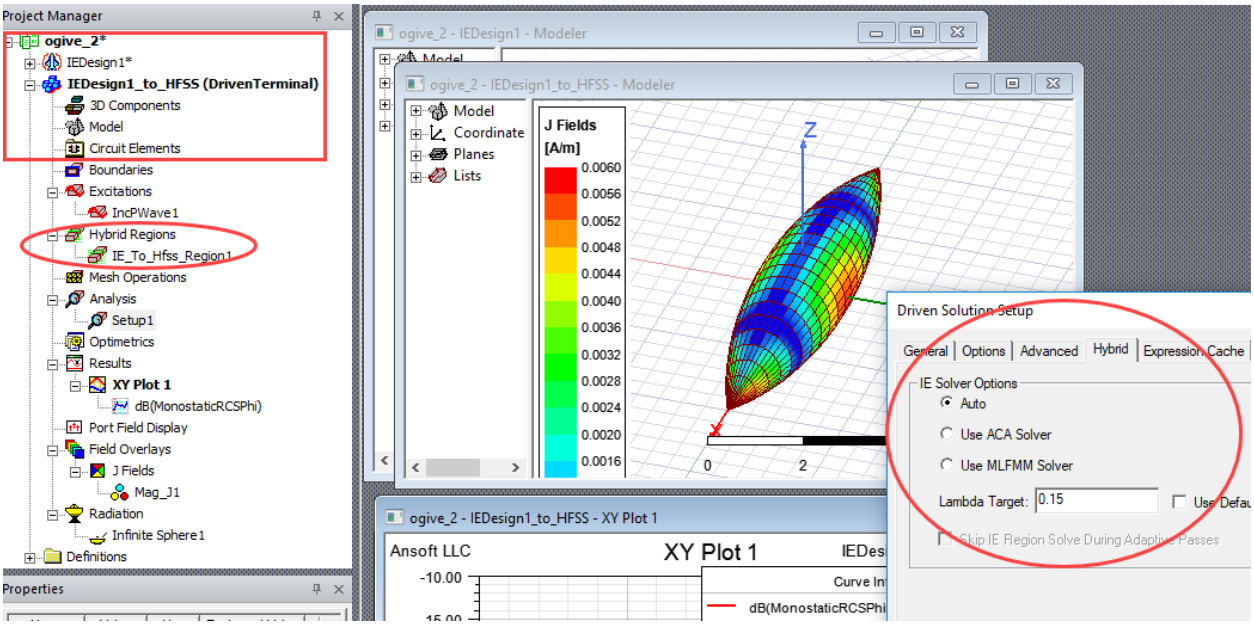
Opening Example Projects

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 Legacy HFSS-IE Projects

If you open an existing HFSS-IE project in Ansys Electronics Desktop, it is automatically converted to an equivalent HFSS project with IE Regions assigned.



The primary changes in the converted project are that an IE Region is assigned to the objects, and that the **Hybrid** tab of the Setup has the IE Solver Options. All Excitations, Plots, Field Overlays, and Radiation are all equivalent to those in the original project. If you select the original project, the HFSS-IE menus will still appear for that project. However, going forward, HFSS-IE will not be supported. IE Regions should be used.

HFSS-IE included a Physical Optics (PO) solver for use with metallic designs excited by an incident plane wave, a far field wave, or a near field wave. The PO functionality can be achieved using PO Regions. This solver provides first-order scattering information, such as an approximate RCS. If that result is not accurate enough (for example, for a back lobe calculation), you can use the regular IE solver. The types of reports are the same; it is the accuracy that changes. When selected, it solves for only one pass, performing no adaptive refinement.

Opening Legacy Projects

To open a legacy project:

1. Click **File** > **Open**.
2. From the **Look in** drop-down menu, select the location of the project. In the folder list, double-click folders to find the one that contains the project.
3. Double-click the project 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.

Legacy Project Translation

When you open a legacy project, virtually all of the project's preprocessing data is translated. Solution results and Optimetrics setup data are unavailable; however, the nominal model created for Optimetrics is translated.

The following table contains additional notes about the translation of legacy project information.

Model Geometry	• The translated geometry's construction history is unavailable; therefore, the original object properties you defined cannot be modified in the **Properties** window. • For units unavailable in the older version, such as yards, the nearest available units are used; the model will be scaled slightly to fit the new units. • View visualization settings apply to the saved design. If these have been changed from the default (15 deg), this affects the memory and CPU required to open the project.
Excitations and Boundaries	• If the legacy project contained both port impedance and calibration lines, the impedance lines are translated to integration lines, and calibration lines are ignored. If the project contained both impedance and terminal lines, both are translated to integration lines. The impedance lines are ignored for Driven Terminal solutions and terminal lines are ignored if the project is changed to a Driven Modal solution. • Boundaries assigned to named interface selections or rectangle selections are not translated.
Hybrid Regions	• Radiation Boundaries with FEBI become Hybrid Regions assigned as FEBI.
Materials	• Functions defined in legacy projects become project variables; therefore, functional material properties are translated. • Perfect conductors become regular materials with conductivity values of 1E30. • Object coordinate systems are created for objects assigned anisotropic materials in legacy projects. The coordinate system is defined at the same origin as the global coordinate system with the same orientation defined when the anisotropic material was assigned to the object in the legacy project. • Nonlinear materials from legacy projects that have magnetic saturation values greater than zero are treated as ferrite materials. Their properties are not modified.
Mesh Operations	• Mesh refinement operations performed on arbitrary boxes in legacy projects are ignored.

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

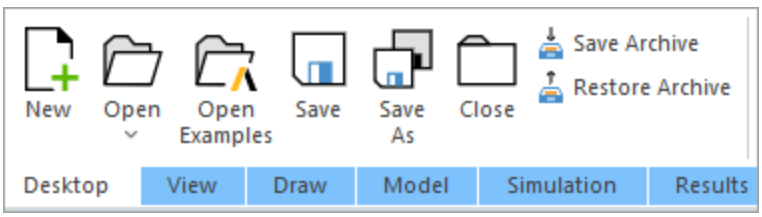
	• Area and volume-based mesh operations are translated as length-based mesh operations by taking their square roots and cube roots, respectively.
Optimetrics	• Setup information, including design variables, is not supported; however, the nominal model can be translated. • Parameterizing a translated model is limited because geometry construction history is unavailable.
Solution Types	• Driven solver projects that contained terminal lines are translated to Driven Terminal solution types.
Solution Setup	• Impedance-only and emissions-only solutions are not supported; therefore, these selections in legacy projects are ignored. • The design's initial mesh is used for the solution. Current meshes are not translated. • Saving dominant-only or higher-order-only mode S-matrix entries are not supported; therefore, these mode selections in legacy projects are ignored. • For frequency sweeps, the **Number of Steps** value specified in the legacy project is converted to the corresponding **Step Size** value. • The total number of requested adaptive passes in the legacy project becomes the **Maximum Number of Passes** value. For example, if you request 3 adaptive passes, solve them, and then request 2 adaptive passes, 5 will be the value specified for the **Maximum Number of Passes**.
Solutions	• Solution data is not translated; therefore, you must solve legacy projects again.

Note:

Legacy Circuit (Serenade) projects (**.ssp** extension) do not open correctly from the **File Open** dialog box. Contact Ansys Technical Support for assistance in converting legacy projects. Opening an Ansys Neutral File project (**.anf** suffix) begins with a conversion dialog box. See Importing ANF Design Data.

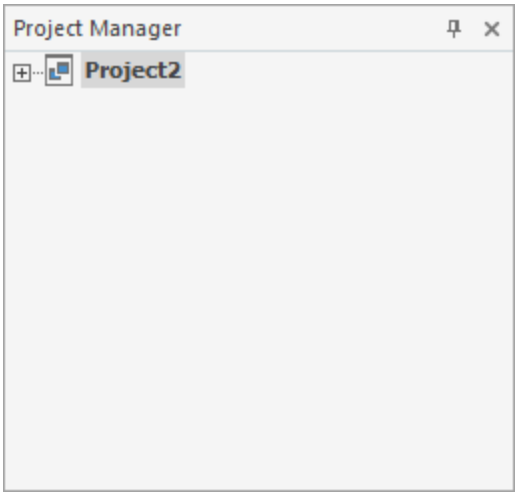
Creating Projects

To create a project, click **File** > **New**, or click the **New** icon on the **Desktop** tab of the ribbon.

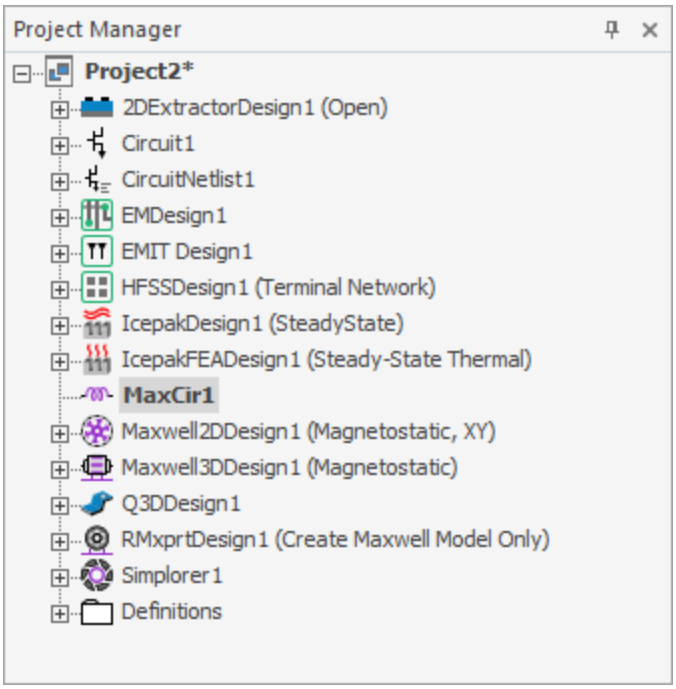


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



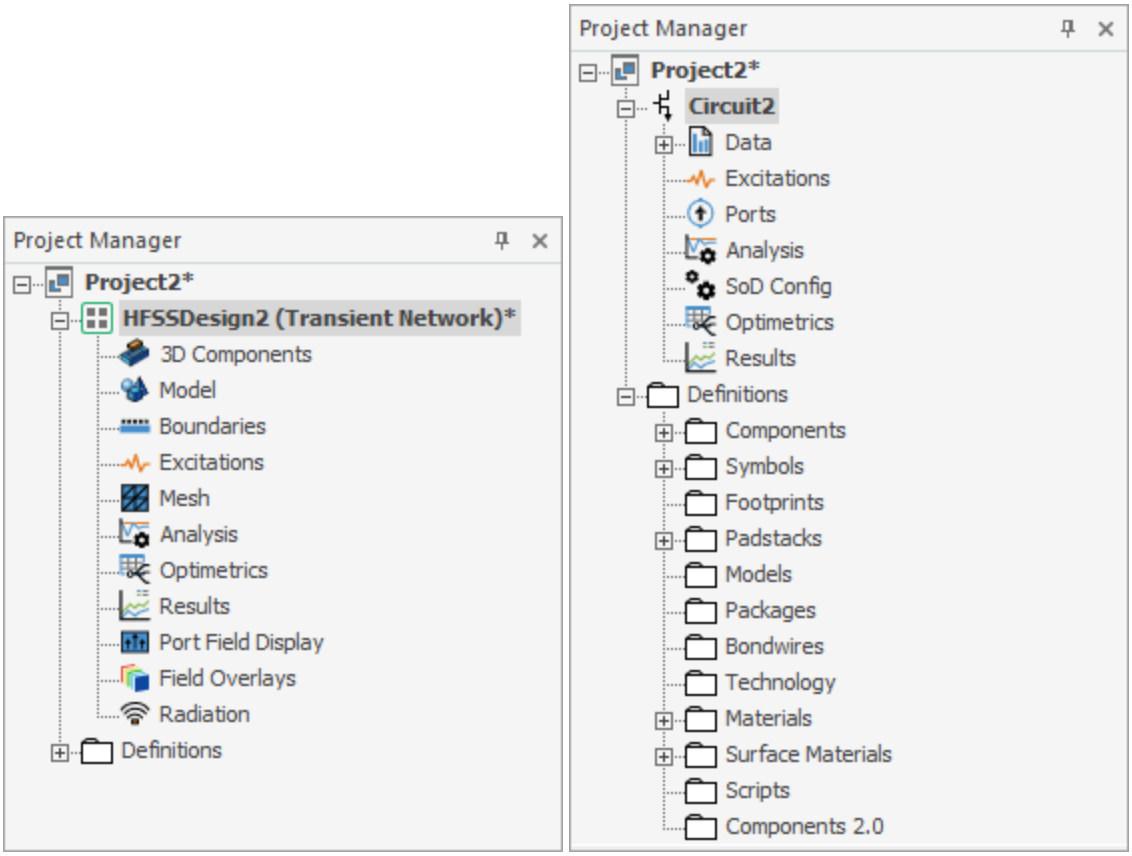
You can insert designs of any type into the project, where they are stored in the project tree.



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.

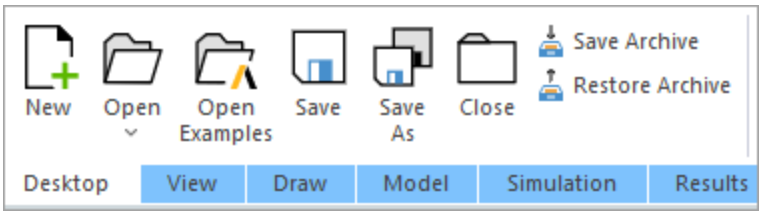


Saving Projects

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.

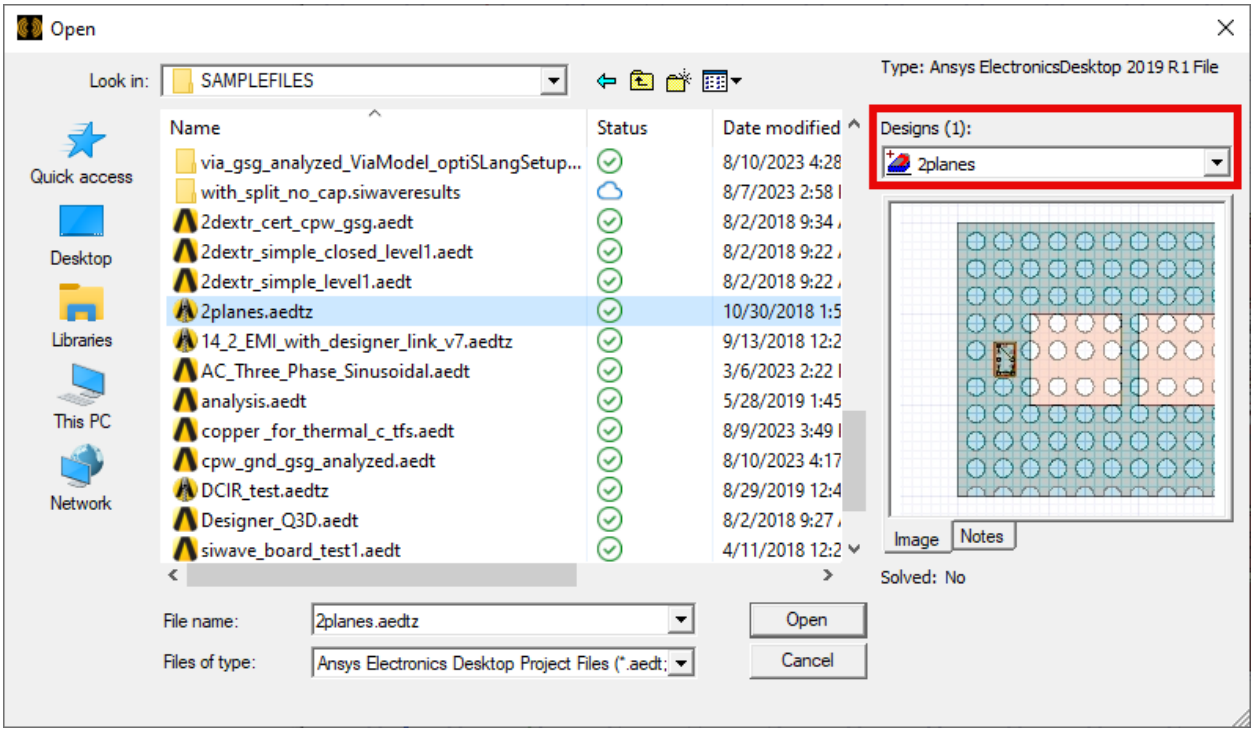
> **Warning:** If you choose to recover the auto-save file, you cannot recover the original project file that has been overwritten; recovering data in an auto-save file is *not* reversible.

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.

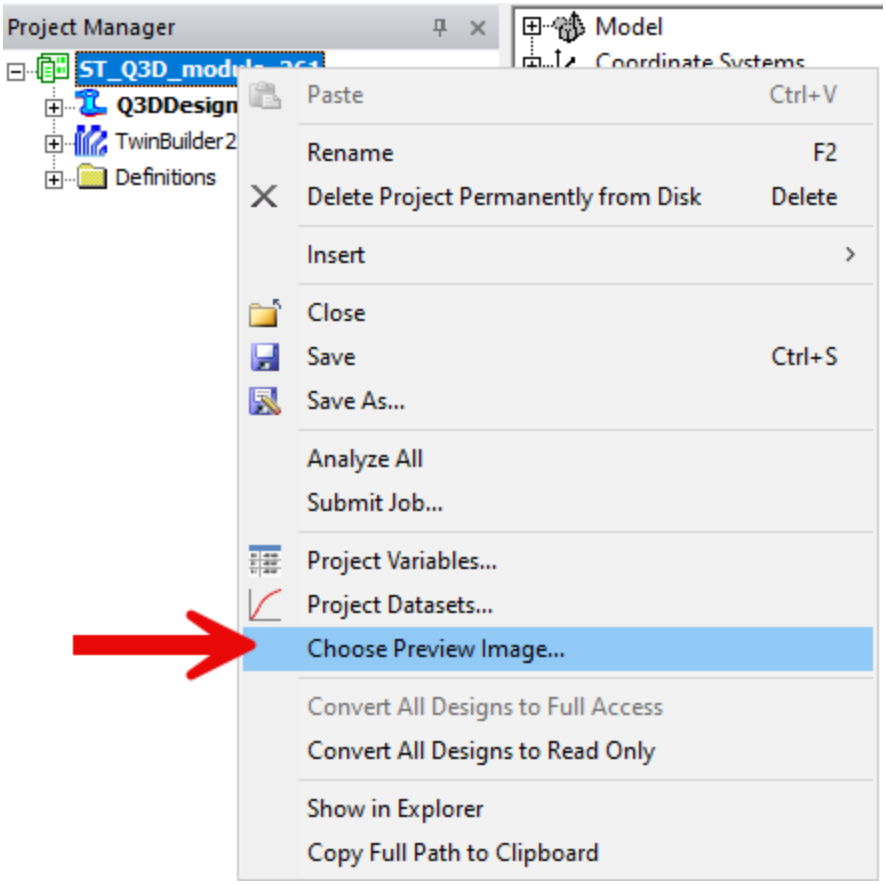
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



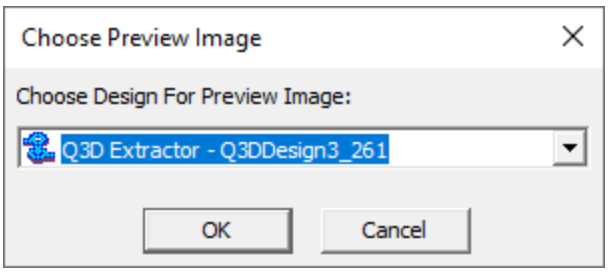
For a project containing multiple designs, users can select which design previews by default:

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.

Copying and Pasting a Project or Design

To **copy** a project or design:

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Copy**.
2. Click **Edit** > **Copy**.

   The project or design is copied for pasting.

To **paste** a project or design:

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Paste**.
2. Click **Edit** > **Paste**.

   The project or design is pasted under the selected project, and an icon is added to the project tree.

> **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,

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

including results files, external files and projects. For example, if a project linked to the main project also has linked or associated files, you can add them.

Archive File Types

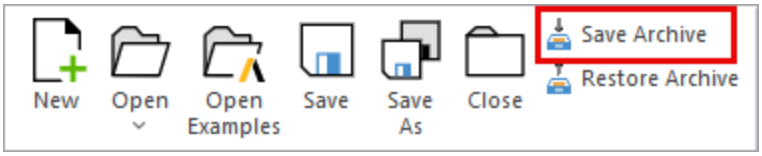
Internally, project archive files are .zip files, and are compatible with any program that can read .zip files. Project archive files have an extension that is unique for each product, and is generated by adding a 'z' to the project file extension (for example, *.aedtz, *.adsnz). This extension displays as the default when saving and restoring archive files.

Archive Preview

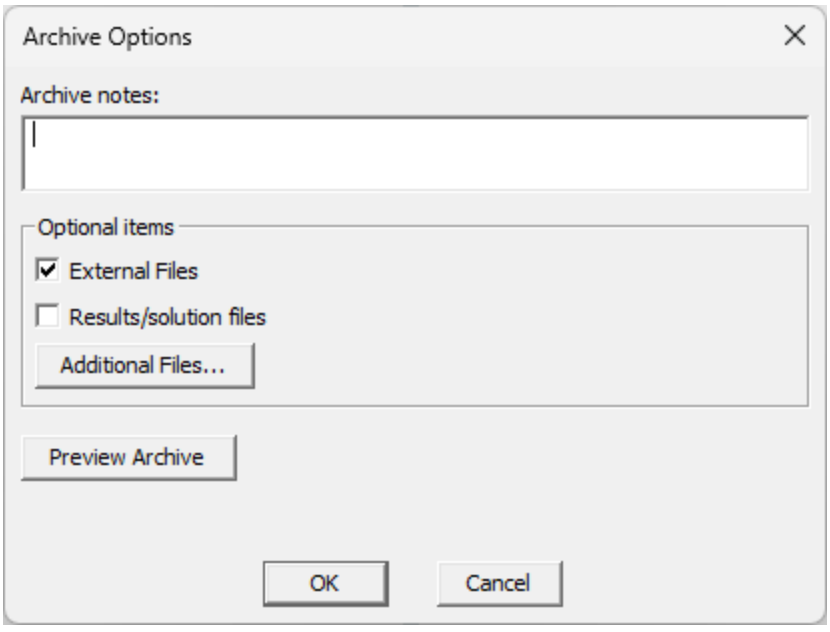
The **Archive** command includes a preview feature that lets you review the contents of a planned archive.

To archive the current project:

1. Click **File** > **Archive** or select the **Desktop** tab of the ribbon and click **Save Archive**.

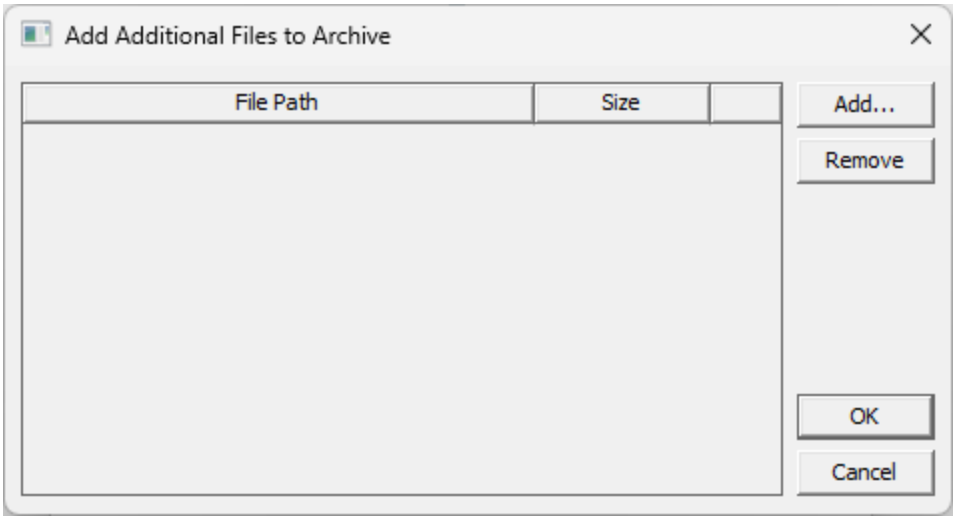


The **Archive Options** dialog box opens.



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

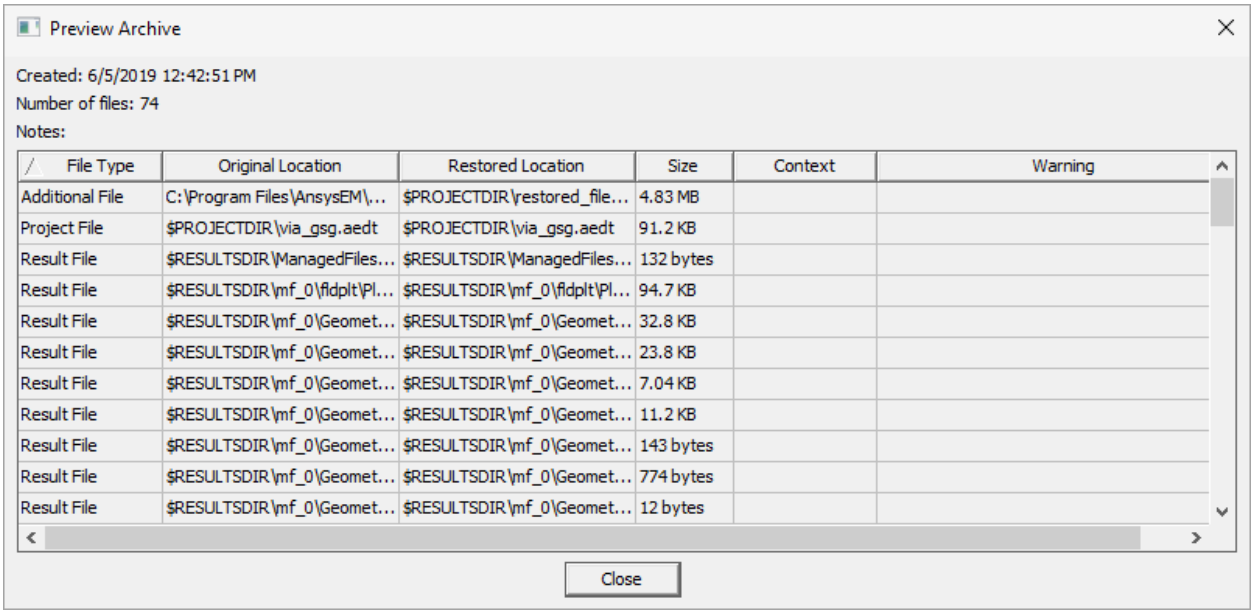
- **Archive Notes** – in this field, you can specify notes that will be visible when previewing the archive. These notes can be viewed from the preview dialog box without restoring the archive.
- **External Files** – selecting this check box causes all external files to be included in the archive. These include any existing files associated with the project, such as linked files, or files added through the **Project** > **Insert Documentation File** command or **Project** > **Data Set** command.
- **Results/Solution Files** – selecting this check box causes the entire results directory to be included in the archive. This may greatly increase the size of the archive file.
- **Additional Files** – clicking this opens the **Add Additional Files to Archive** dialog box.



  - 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.

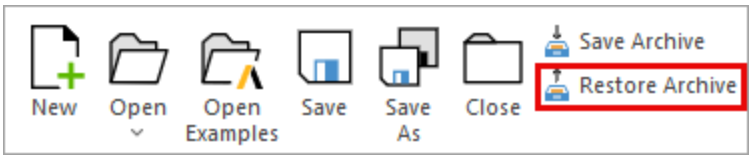
To achieve this, it is sometimes necessary to change the location of files in the archived project so that the external files are located in the project directory. At archive time, any external files not located in the project directory are relocated to the restored_files subdirectory of the project 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.

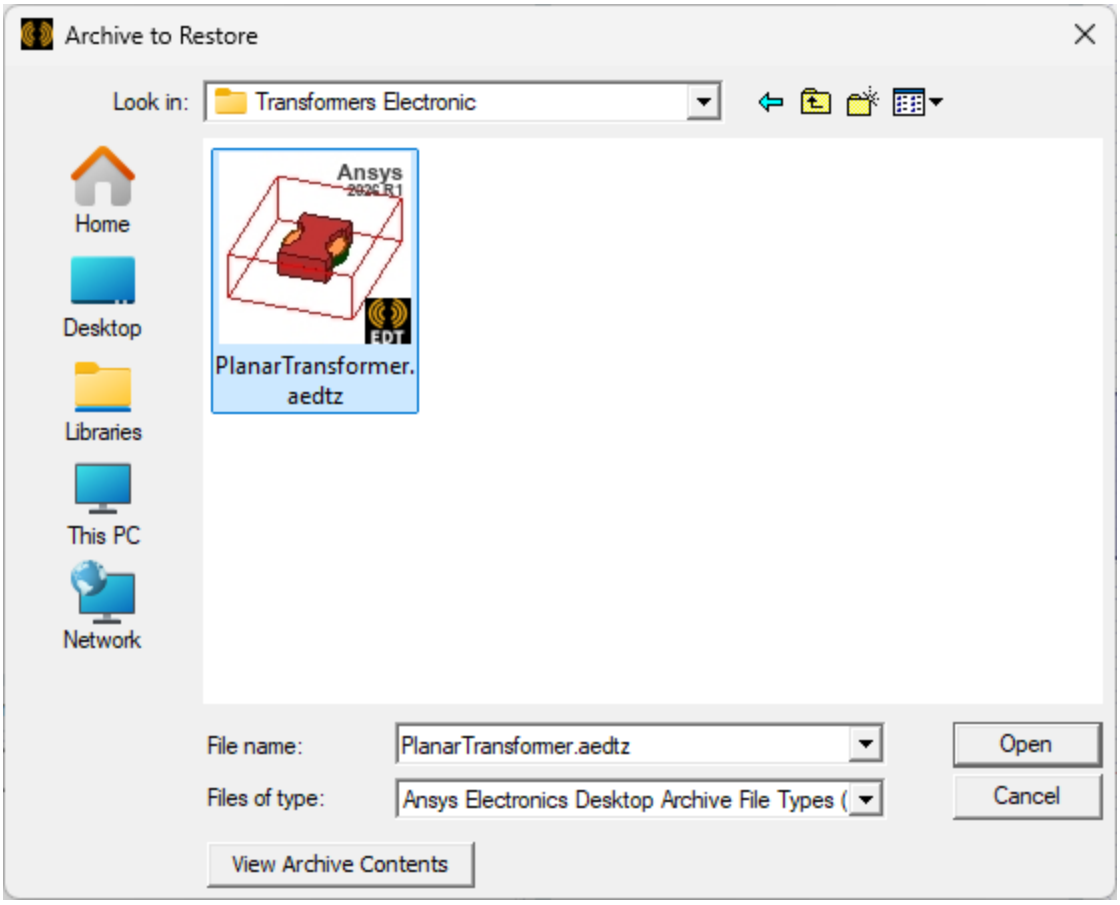
Restoring an Archived Project

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**.



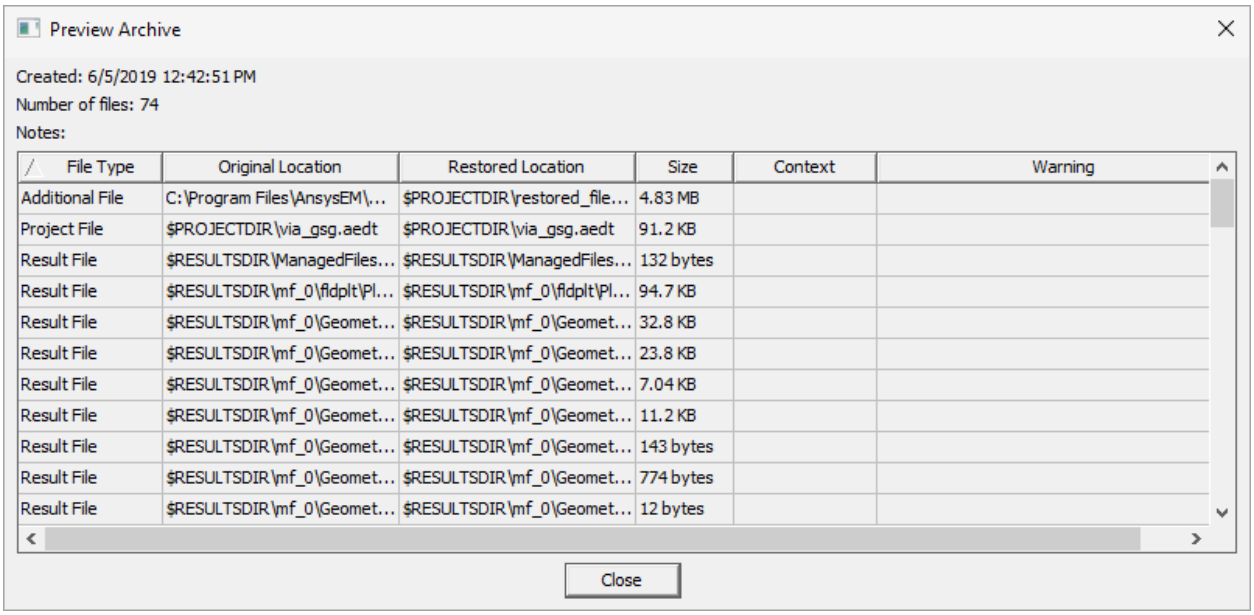
This displays an **Archive to Restore** browser window that lets you navigate your file system for *.aedtz, legacy *.<*solversuffix*>*z* or *.zip archive files.



2. If you select a valid archive file, you can click **View Archive Contents** to preview the contents.

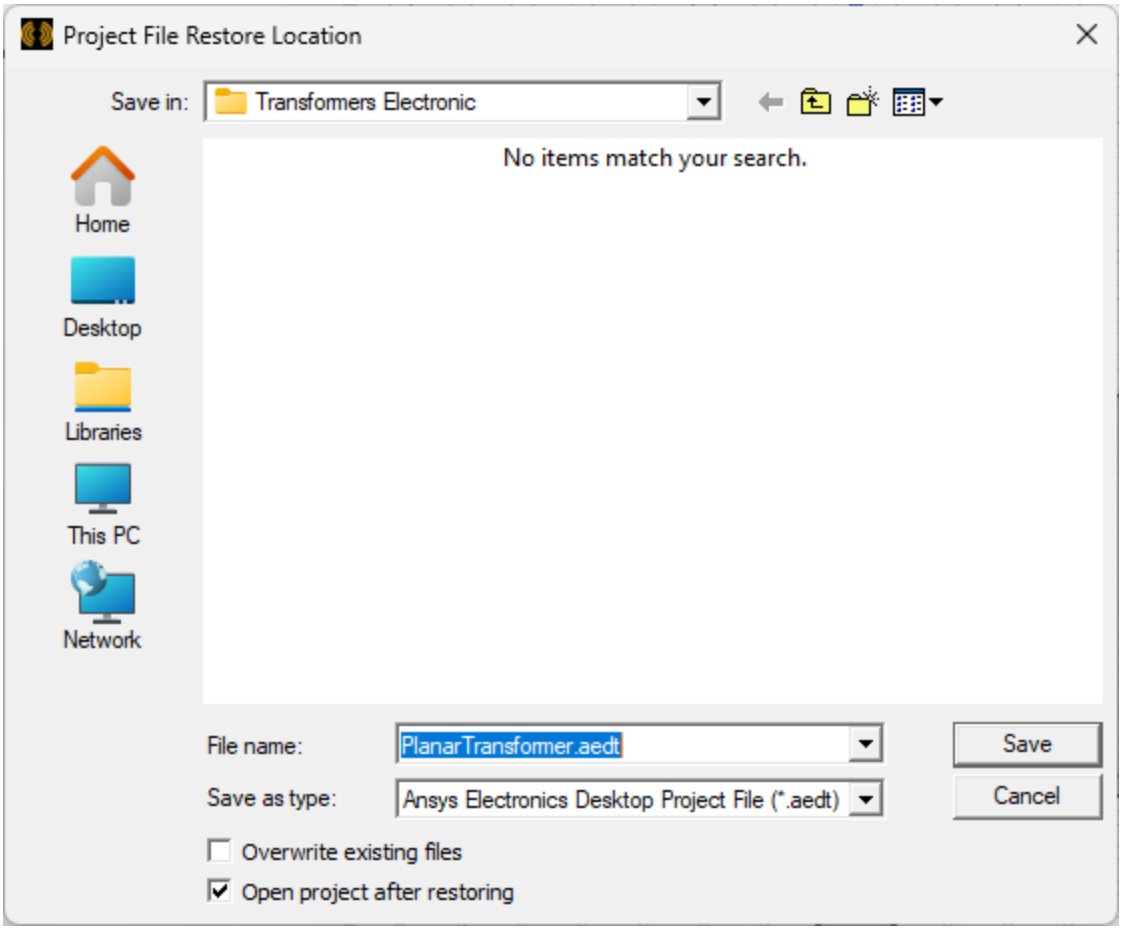
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



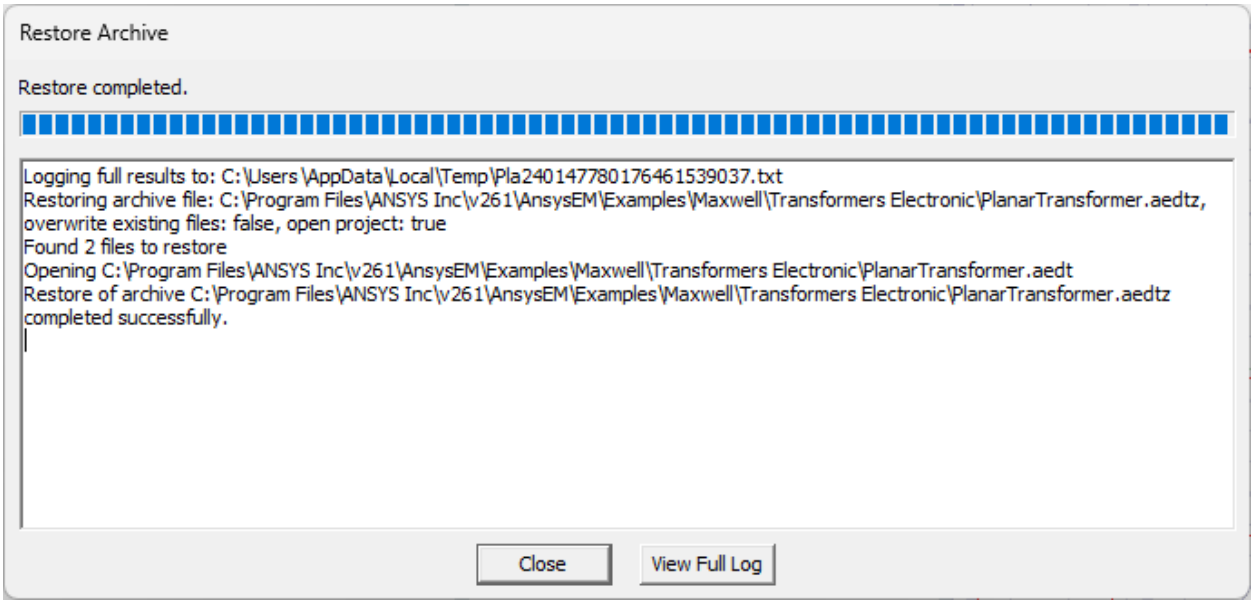
You can edit the file name, and select options:

- **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**.

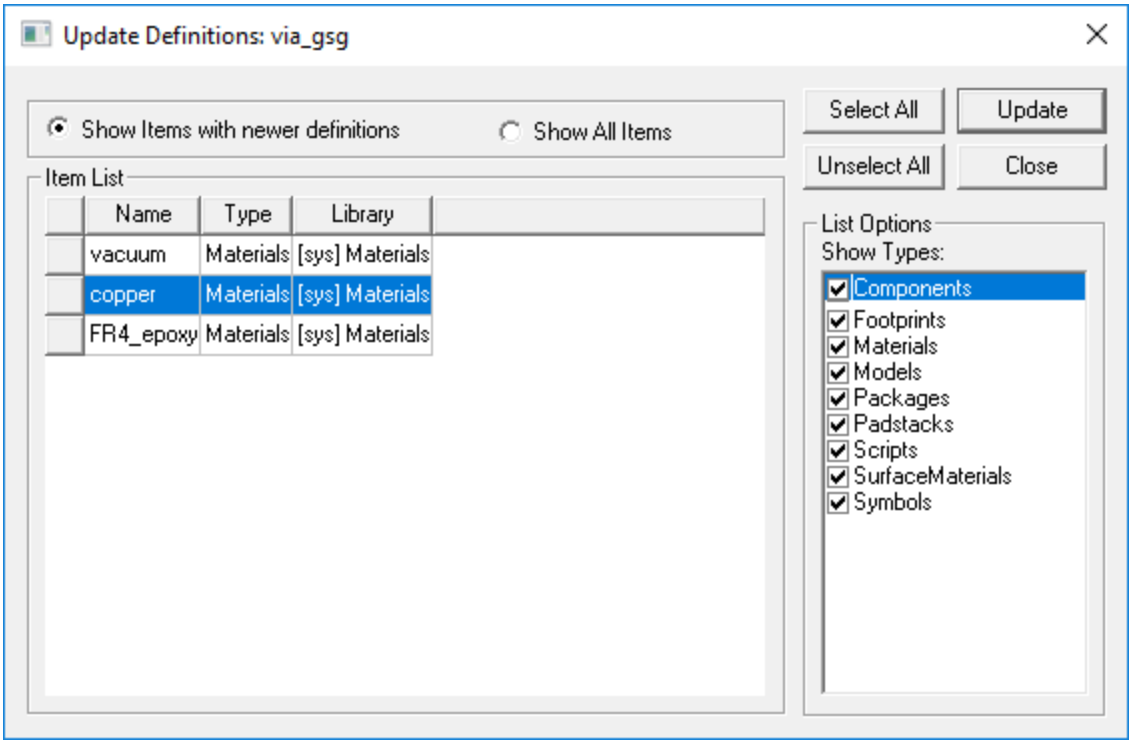
> **Note:**
>
> Read Only settings are not saved when you save the project file.

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.

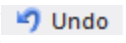
Use the **Redo** command to reperform an action you previously undid.

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 **HFSS 3D Layout** > **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.

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 **HFSS 3D Layout** > **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.

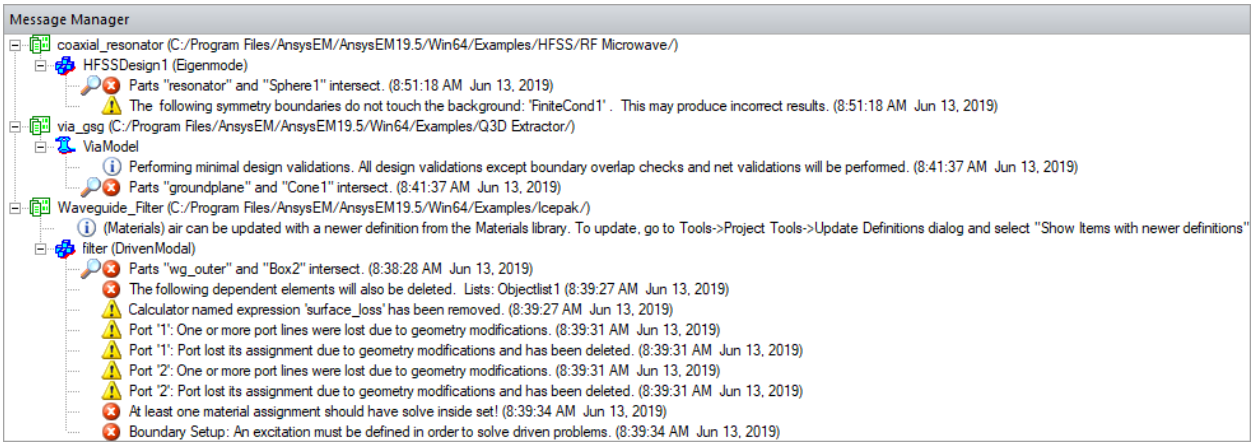
2. View the results of the validation check in the **Validation Check** window.

   The following icons can appear next to an item:

	Indicates the step is complete.
	Indicates the step is incomplete.
	Indicates the step may require your attention.

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

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 **HFSS 3D Layout** > **Validation Check** to run another validation check. Repeat until the Validation Check completes with no issues.
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:

1. Select **HFSS 3D Layout** > **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**.

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

- **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.

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

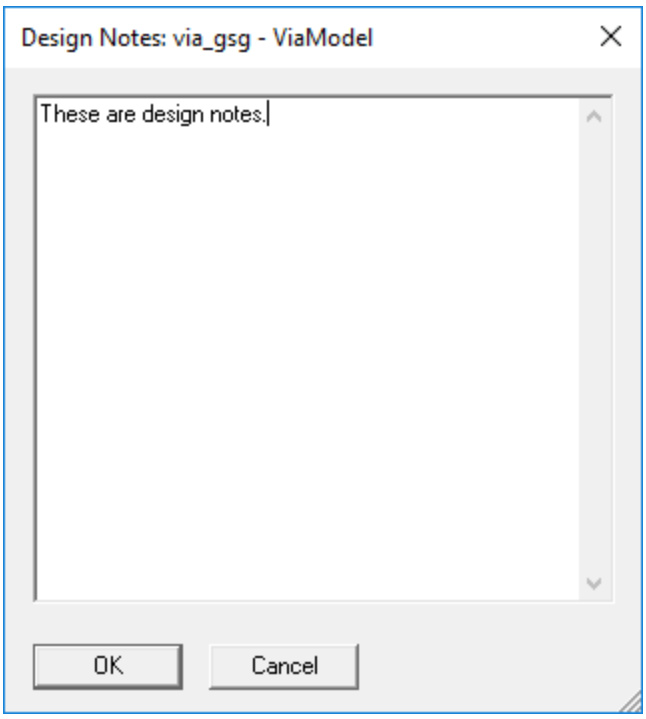
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can 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 **HFSS 3D Layout** > **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.

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.

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

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.

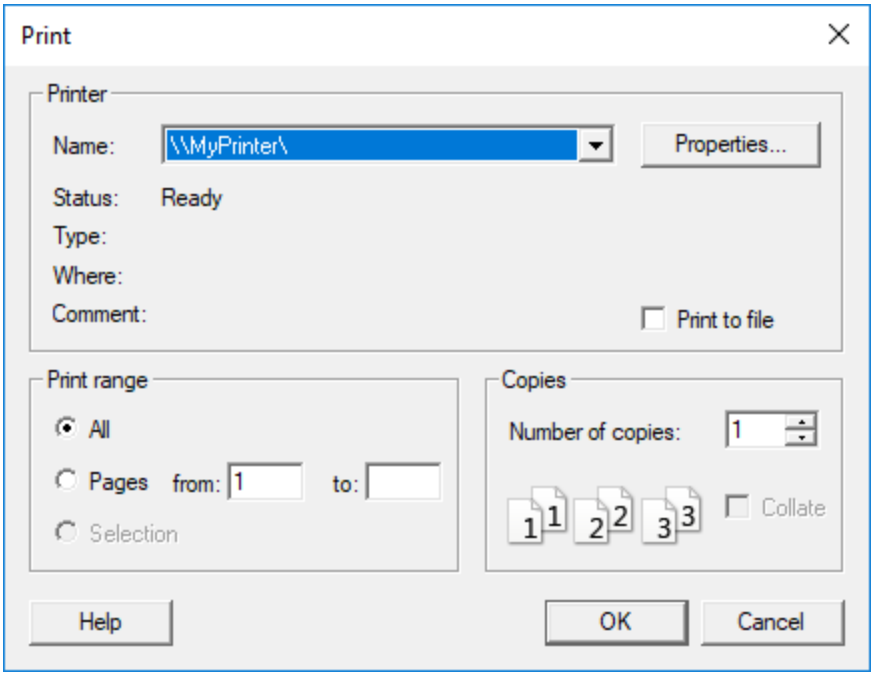
Printing

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:



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**.

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

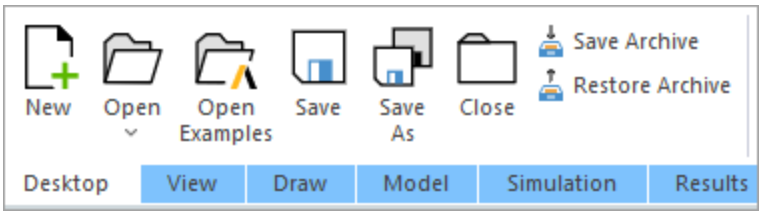
Important:

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.

Closing Projects

To close the current project, select **File > Close** or select the Desktop tab of the ribbon and click the Close icon.

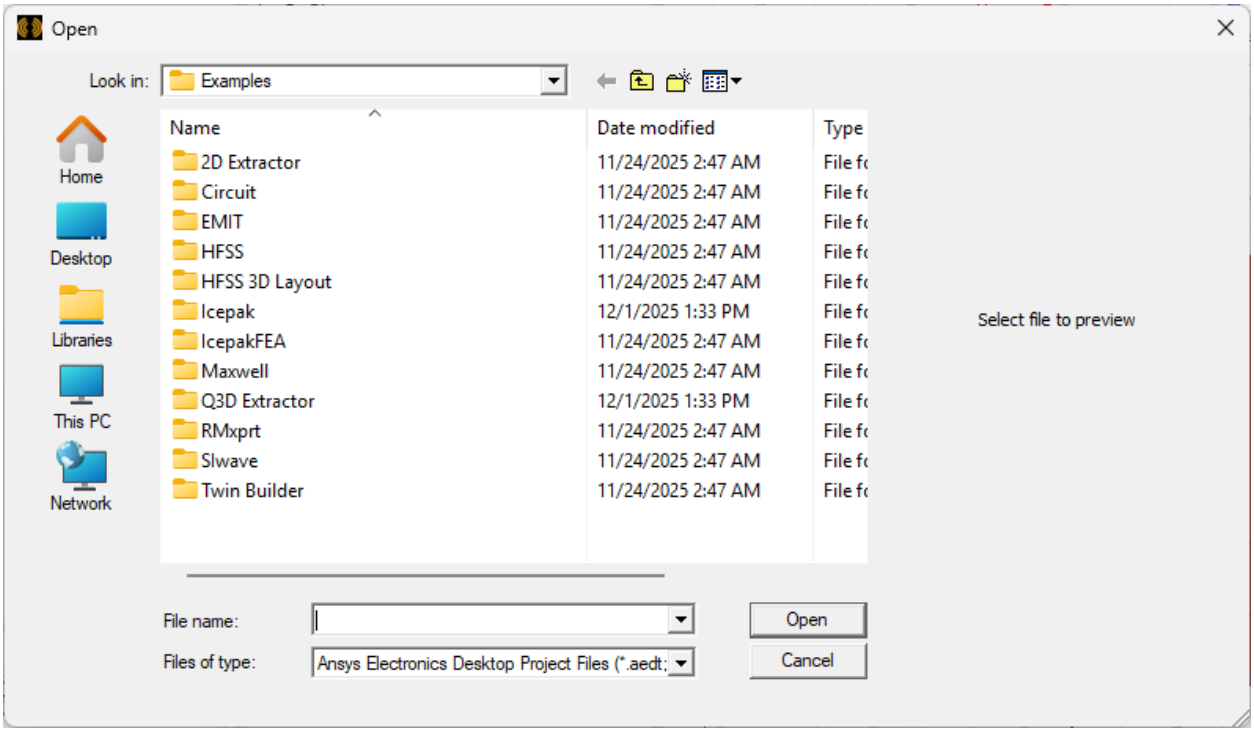


If a simulation is running, a message appears stating that if you continue, the simulation will abort before closing. If you click **OK**, the Desktop aborts the simulation and closes the project.

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. The examples under Circuit, HFSS, and HFSS 3D Layout include subfolders, organizing the projects according to different applications:

- Circuit has subfolders for Automation, High Speed Digital, Low Noise Amplifier, RF Microwave, and Signal Integrity examples. The High Speed Digital folder has further subfolders for PCIe and USB projects. And the RF Microwave folder has further subfolders for Amplifiers, DC-IV, Filters, Misc, Mixers, Oscillators, and System projects.
- HFSS has subfolders for ADAS, Antennas, EMI EMC, Filters, Metamaterial, RCS, RF Microwave, Signal Integrity, and Transmission Lines.
- HFSS 3D Layout has subfolders for Antennas, Component, Filters, FSS, and Signal Integrity.

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

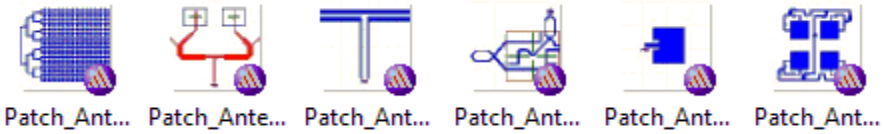
- HFSS 3D Layout Example Projects

HFSS 3D Layout Example Projects

The examples directory for HFSS 3D Layout has subdirectories for Antennas, Component, Filters, FSS and Signal Integrity.

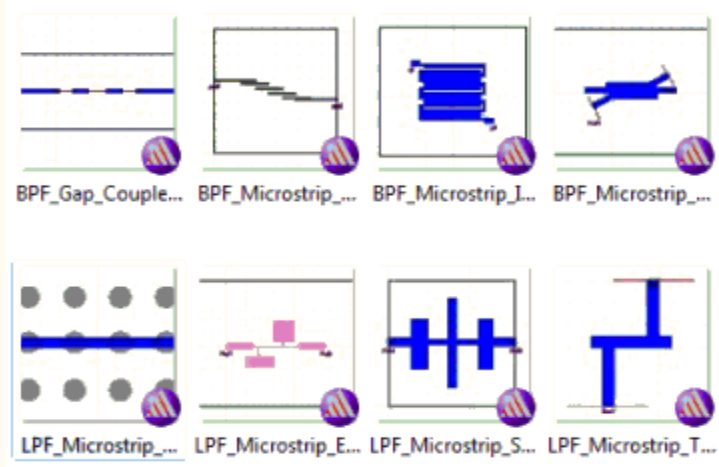
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Antennas

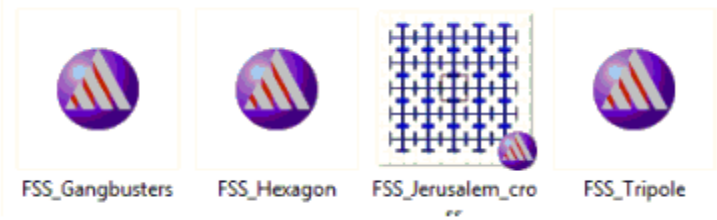


- Components

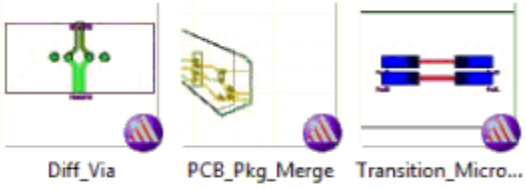
- Filters



- FSS



- Signal Integrity



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

Exporting Options Files

Ansys Electronics Desktop can export user and host options files in XML format.

Exporting Options Files Using the Desktop UI

Export options files by selecting **Tools** > **Options** > **Export Options Files**. This brings up a browser dialog box to select the destination directory.

Click **Open** to copy all config files for the current user and current host to the specified directory. Config files for the install, install_machine, user, and user_machine levels will be copied, if they exist. One additional file, admin.XML, will also be copied to the destination directory; this file does not contain user configurable options.

Exporting Options Files Using a Script

A script command has been added that exports options config files:

ExportOptionsFiles

Syntax	`ExportOptionsFiles <DestinationDirectory>`
Return Value	None.
Parameters	BSTR <DestinationDirectory>
Python Example	`oDesktop.ExportOptionsFiles ('D:/test/export')`

Working with Variables

A variable is a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter in **HFSS 3D Layout**. 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 animate a plot against a variable.

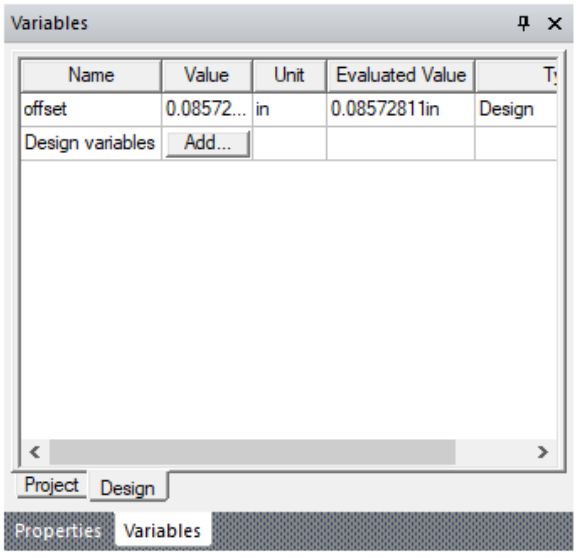
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

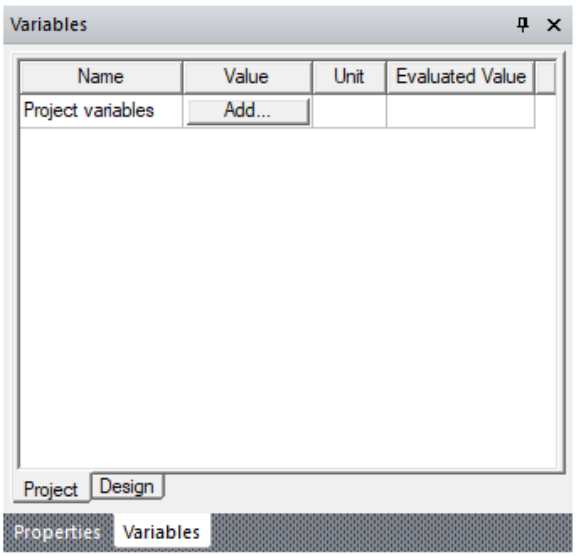
Variables can be regular variables, array index variables, or arrays. A project variable's value can reference other project variables or intrinsics. Values can be a number, variable, or mathematical expression.

There are two types of variables in HFSS 3D Layout:

Project Variables	A project variable can be assigned to any parameter value in the HFSS 3D Layout project in which it was created. HFSS 3D Layout 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 HFSS 3D Layout will automatically append the project variable's name after you define the variable. Project variables can be designated as Design, ArrayIndex or Separator variables .
Design (Local) Variables	A design variable can be assigned to any parameter value in the HFSS 3D Layout design in which it was created. From the Design Variables **Properties** dialog box, you can Add, Add Array, Edit, or Remove design variables. Design Variables can be designated as Design, ArrayIndex or Separator variables

Clicking **View** > **Variables** brings up a dockable variable window that is associated with the active project and/or design. When there is an active project, there will be a corresponding project variable tab. When there is an active design, there will be a corresponding design variable tab. Each tab contains an **Add...** button allowing creation of new variable of this type. If variables exist for the Project or Design, they are shown in the corresponding tab.





Ansys Electromagnetics Suite 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 HFSS 3D Layout project in which it was created. HFSS 3D Layout 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 HFSS 3D Layout 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.

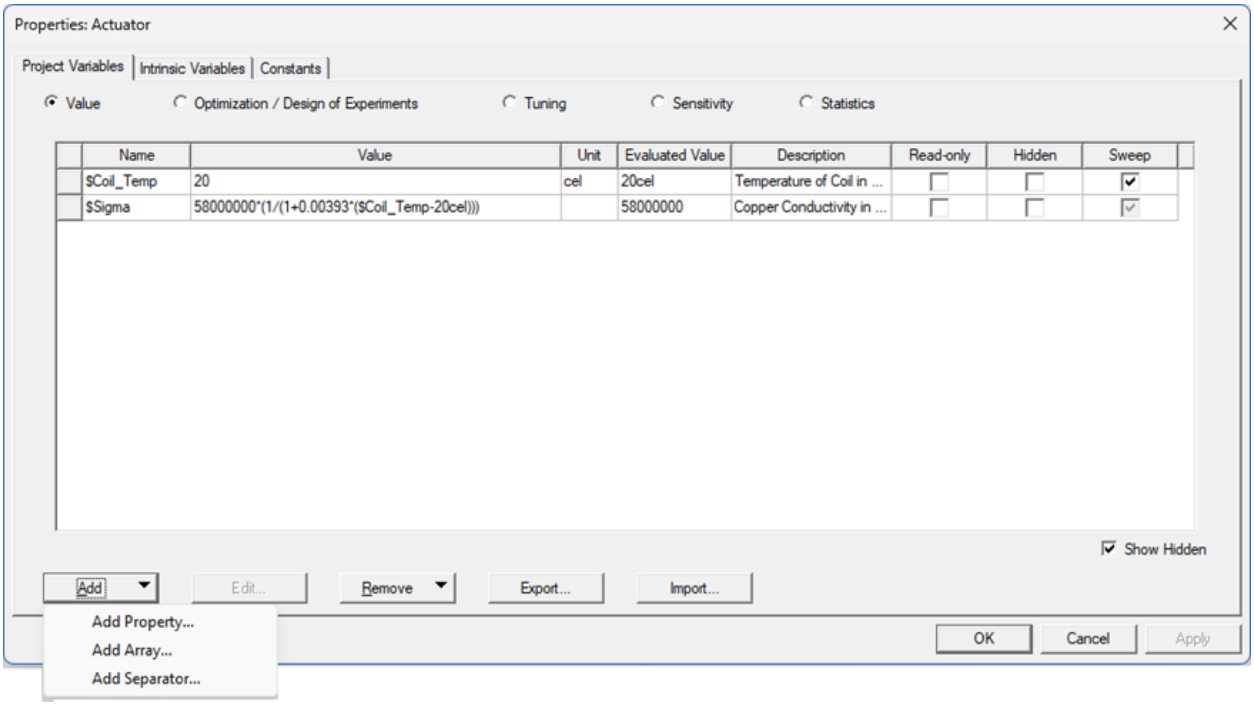
Adding a Project Variable

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 **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Project Variables**.

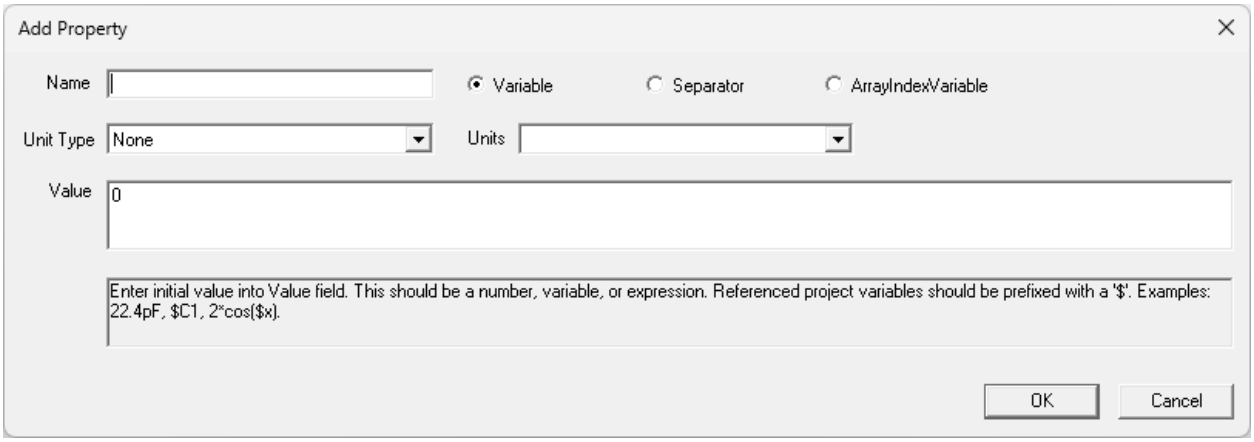
   The **Properties** window appears, on the **Project Variables** tab.

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



2. From the **Project Variables** tab, click **Add Property**.

   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** – use the **Variable**, **Separator**, and **ArrayIndexVariable** 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.

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.

  > **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 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.

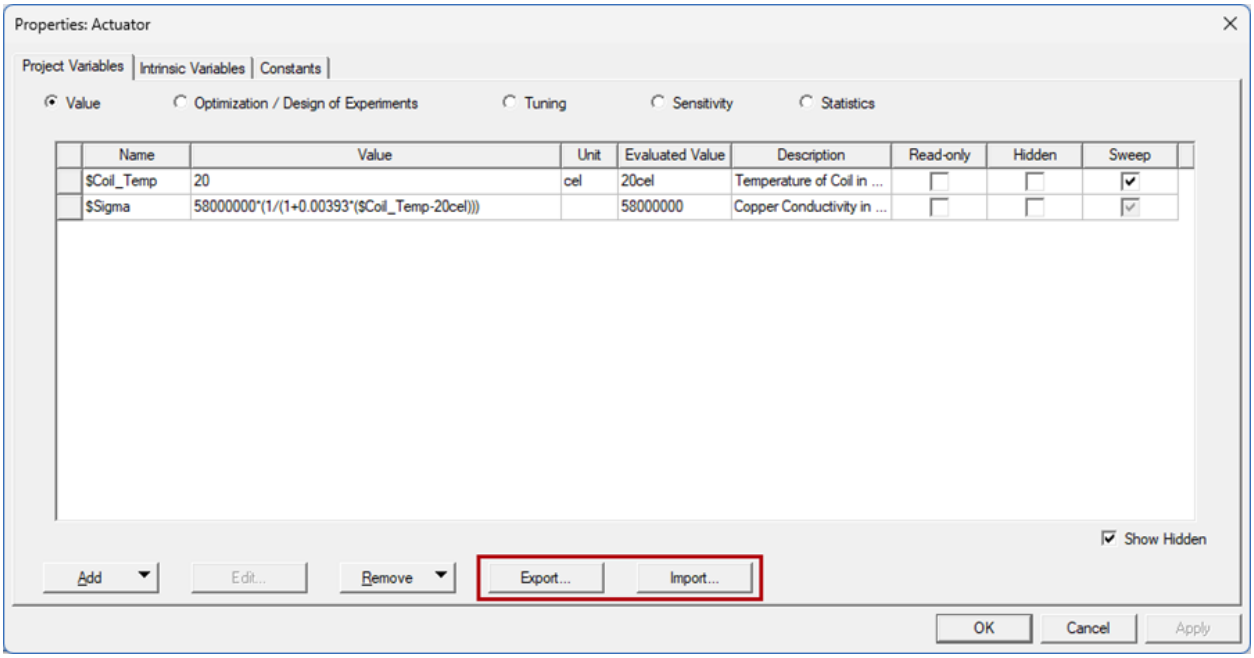
5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show 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.

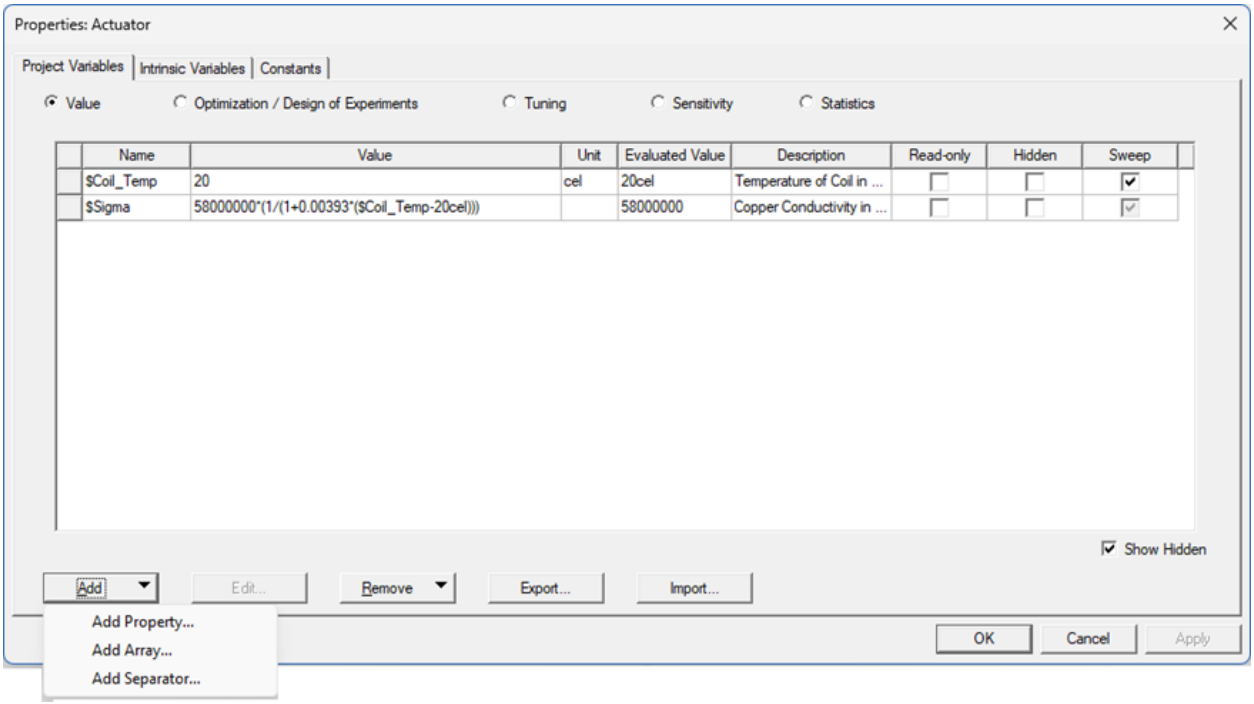
> **Note:**
>
> Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.
>
> The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 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.

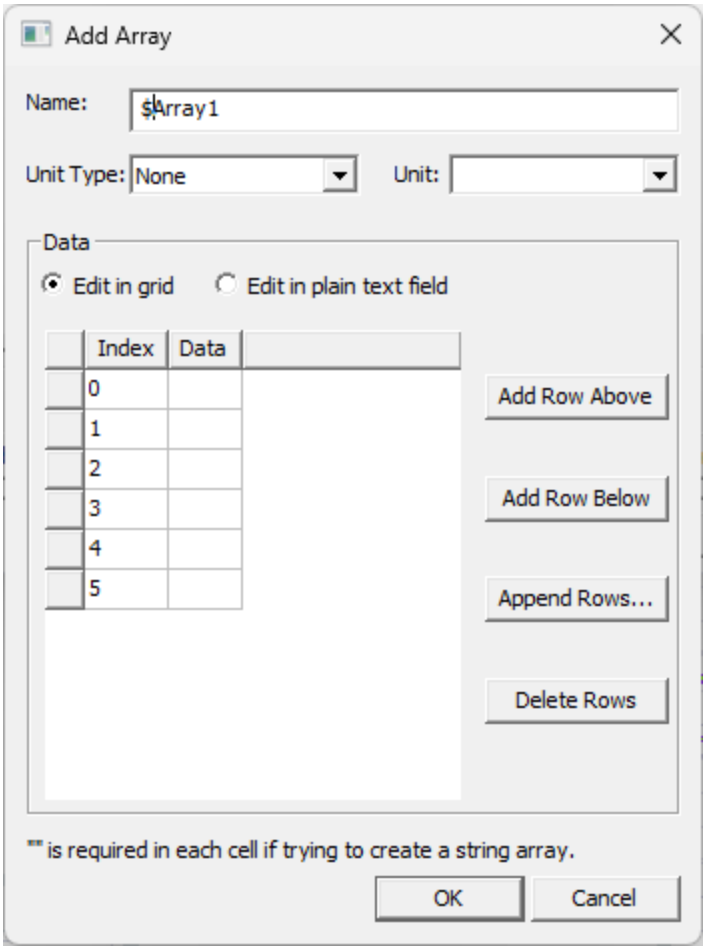
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

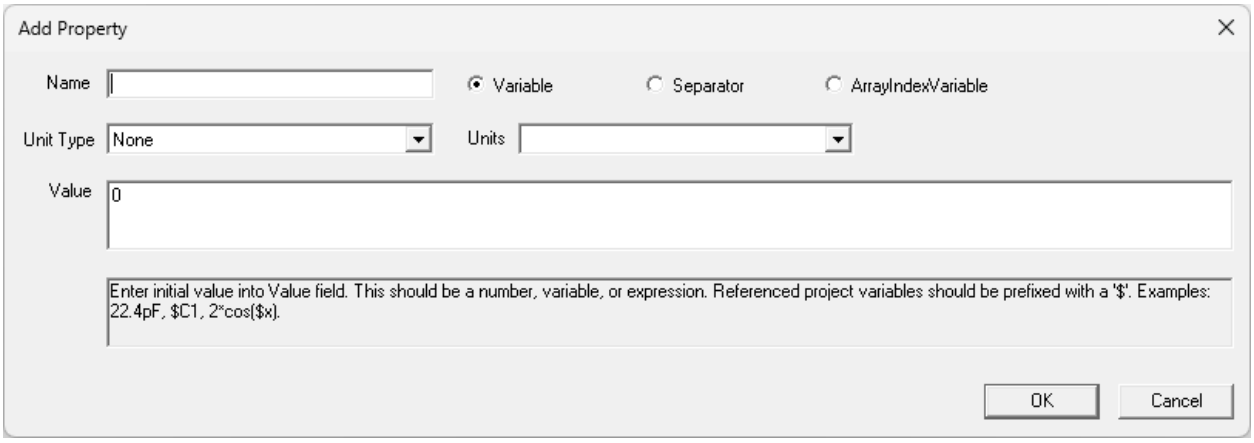
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 HFSS 3D Layout 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:

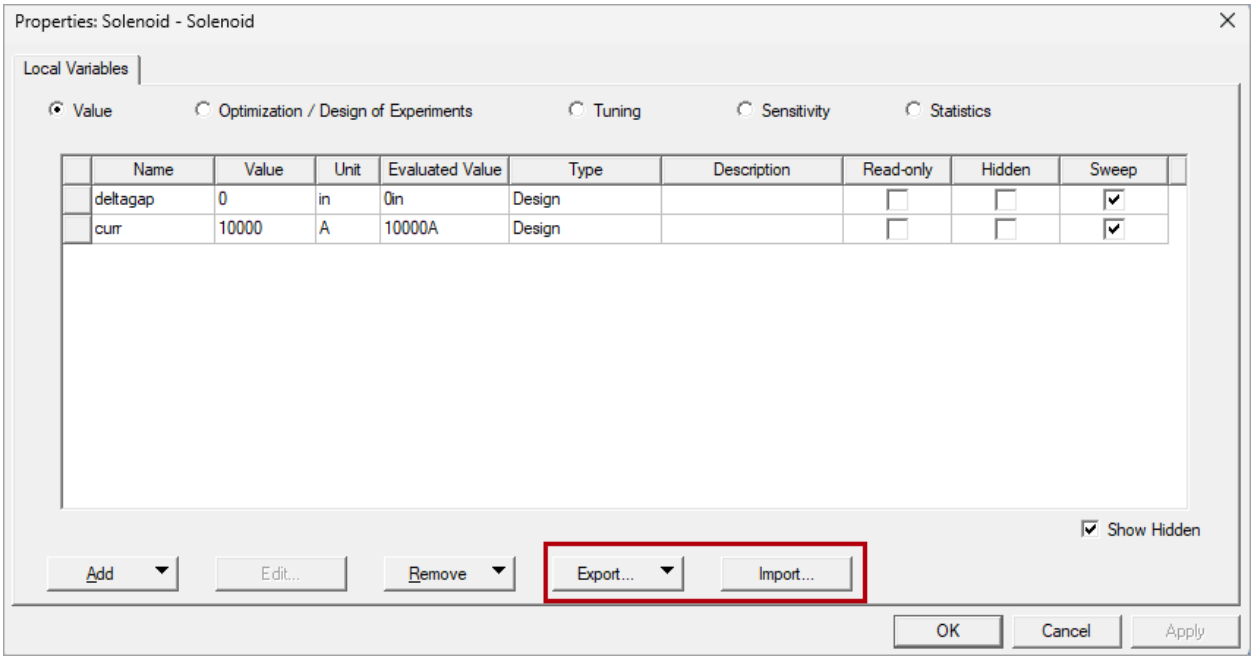
- **Parameter Defaults** – for circuit components and 3D layout components, the Parameter Defaults tab lists local variables with default values that can be overridden in instances of a design. For example, if three subcircuit instances contain a parameter default C1 that is defined as equal to 11.3pF, C1 may be overridden as 11.8pF in the first instance, overridden as 10.9pF in the second, and left at its default value of 11.3pF in the third. A property value that has been set by means of a parameter default is called a *passed parameter*. Parameter defaults can be defined at the design level.
- **Local Variables** – variables, which you define, that apply to the current design only. Local variables can be defined at the design level.
- **General** – predefined design variables that cannot be changed. You cannot use the names of these variables when creating a new variable.

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 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.



Defining Local Variables at the Design Level

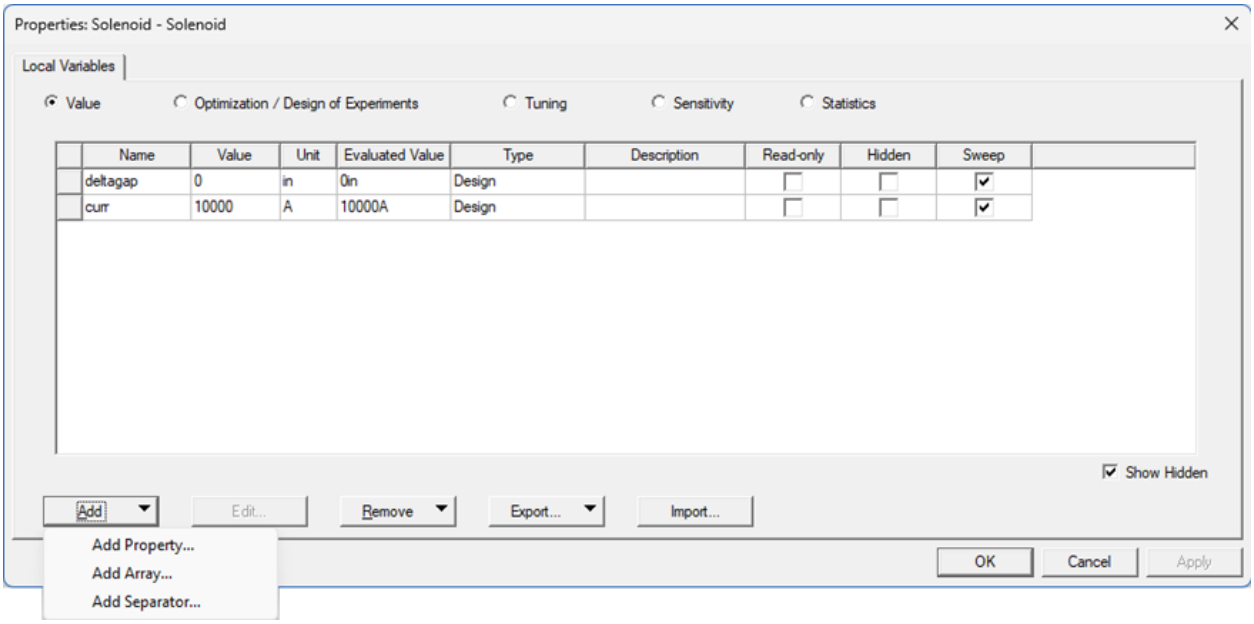
Adding a Local Variable

To define a local variable:

1. Access **Design Properties** one of three ways:
   - Click **HFSS 3D Layout** > **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**.

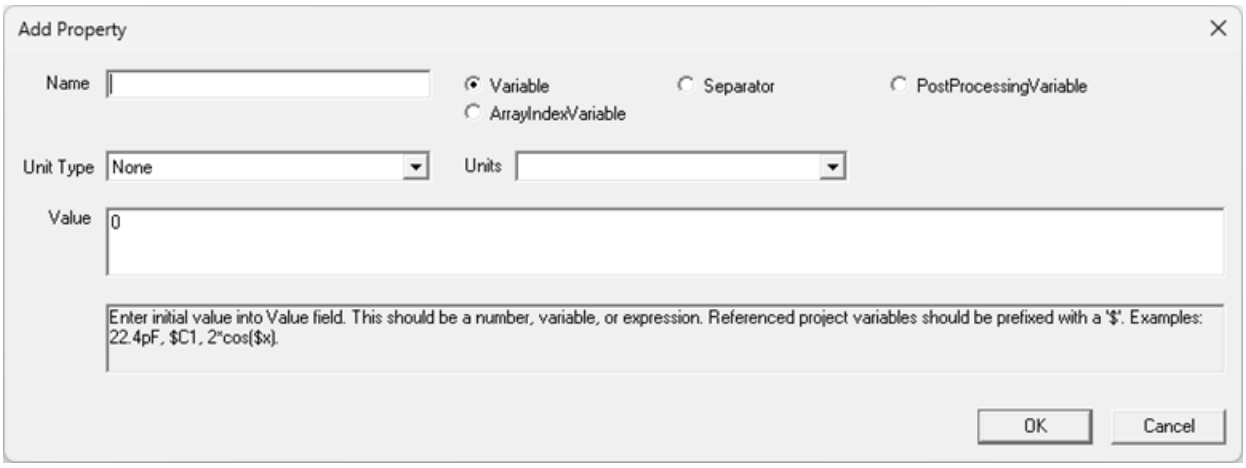
   The **Properties** window appears with the **Local Variables** tab.

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.

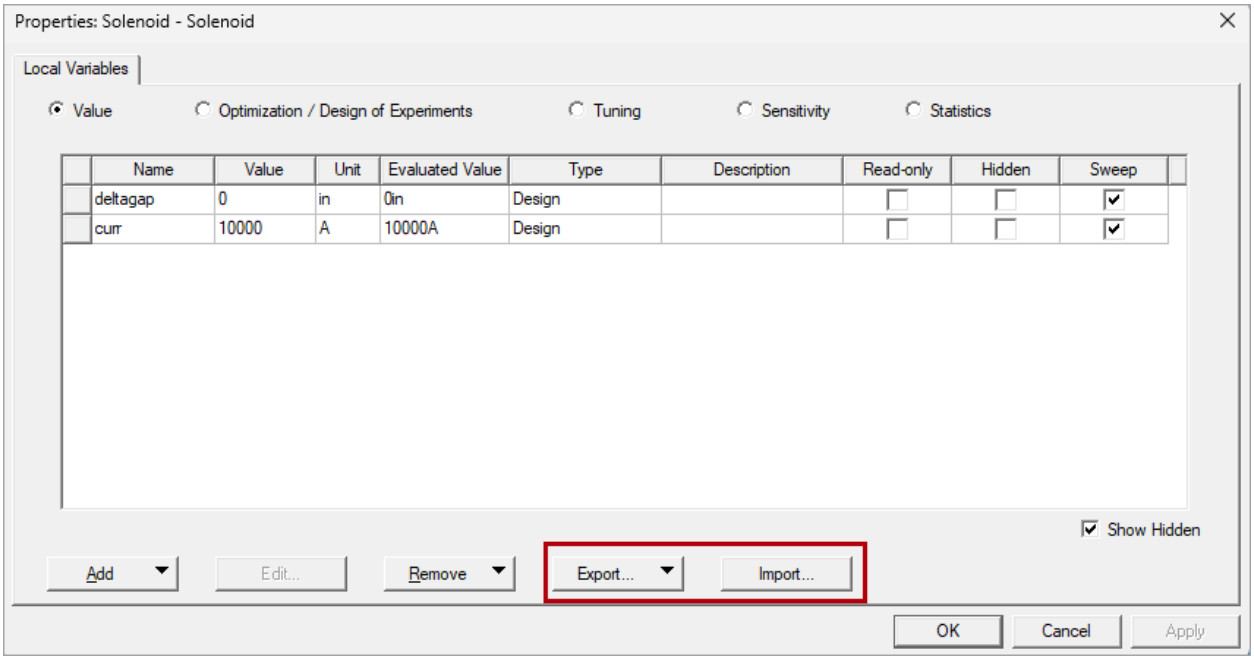
5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show 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

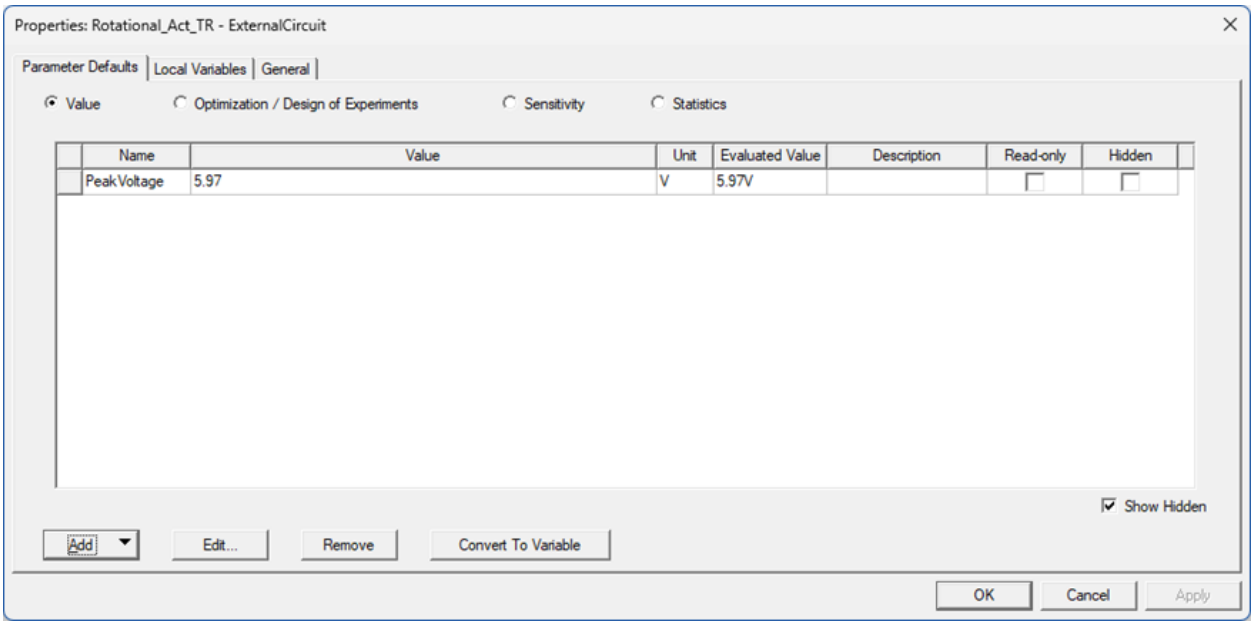
Defining Parameter Defaults at the Design Level

Adding a Parameter Default

To define a new parameter default at the design level:

1. Access **Design Properties** one of three ways:
   - Click **HFSS 3D Layout** > **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.**

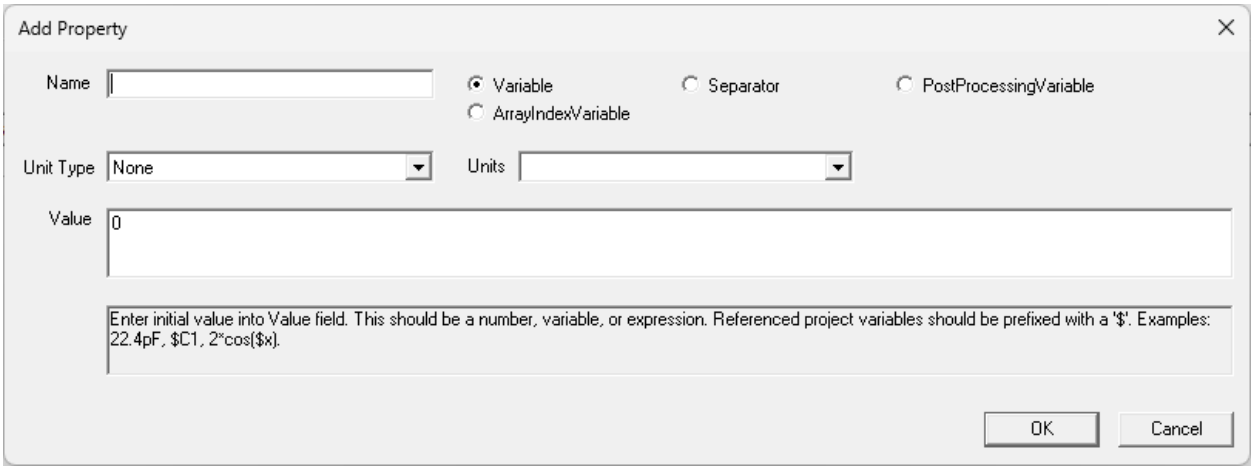
   The **Properties** window appears with a **Parameter Defaults** tab.



2. Click **Add** > **Add Property**.

   The **Add Property** 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.



3. Enter information for the parameter default, as applicable:
   - **Name** – the name can contain alphanumeric characters and underscores (_). You cannot use any names already defined on the **General** tab.
   - **Variable Type** – select **Variable** to enter a value, or select **Separator** to provide a bolded name for a blank line.

     Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.

   - **Unit Type** – for variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).
   - **Units** – for variables, use the drop-down menu to select a unit of measure.
   - **Value** – for variables, enter a valid numeric quantity. Alternately, you can assign a variable to a parameter by typing a variable name or mathematical expression in place of a parameter value.
4. Click **OK**.

   The new variable appears in the list. You can sort local variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.

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

- **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.

6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

Deleting a Parameter Default

To remove a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select a parameter from the list and click **Remove**.

Editing a Parameter Default

To edit a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select the parameter you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Overriding a Parameter Default

You can override a parameter default from an 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.

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 HFSS 3D Layout design for which they 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.

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

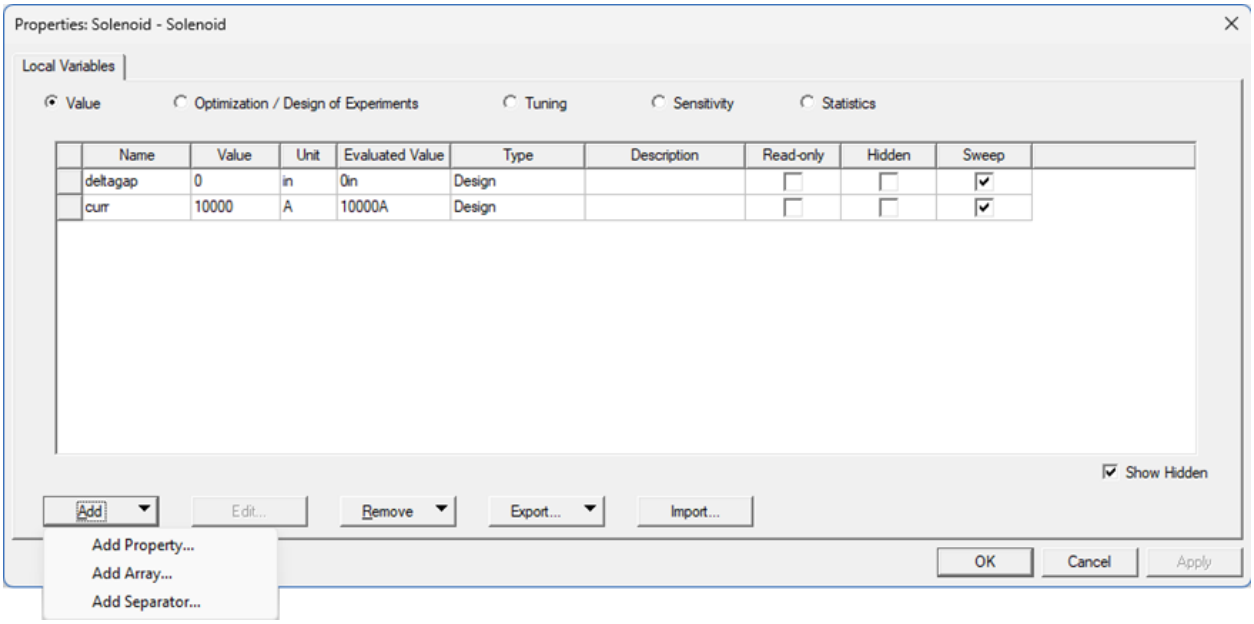
To define an array variable, you must first define a Local Variable Array:

1. Access **Local Variables** one of three ways:
   - Click **HFSS 3D Layout** > **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**

   The **Properties** window appears.
2. Select the **Local Variables** tab.

   The **Local Variables** tab displays.

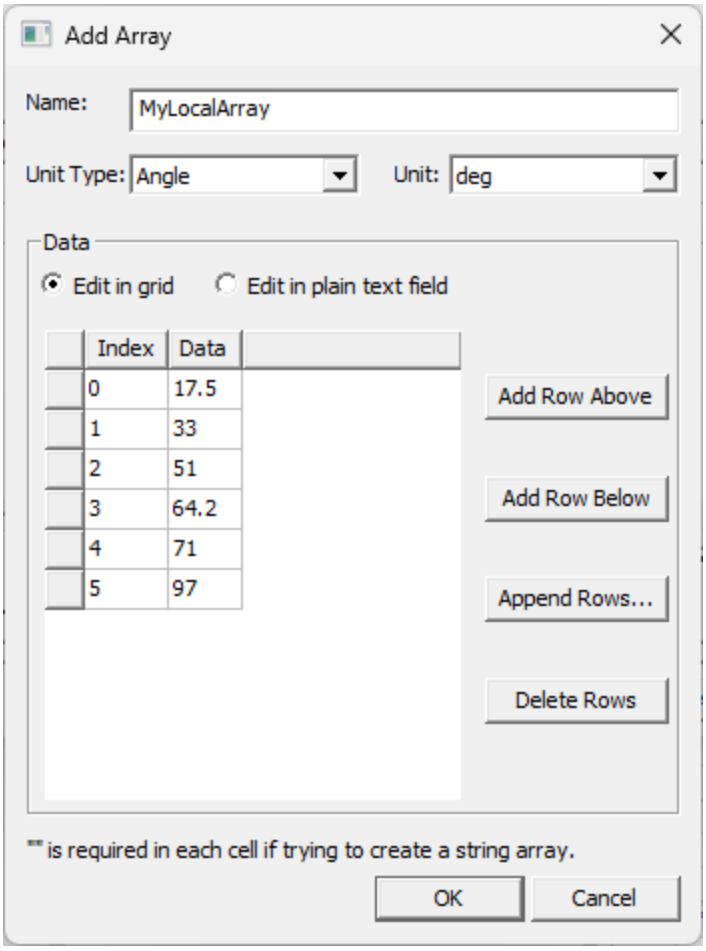
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **Add** > **Add Array**.

   The **Add Array** window appears.

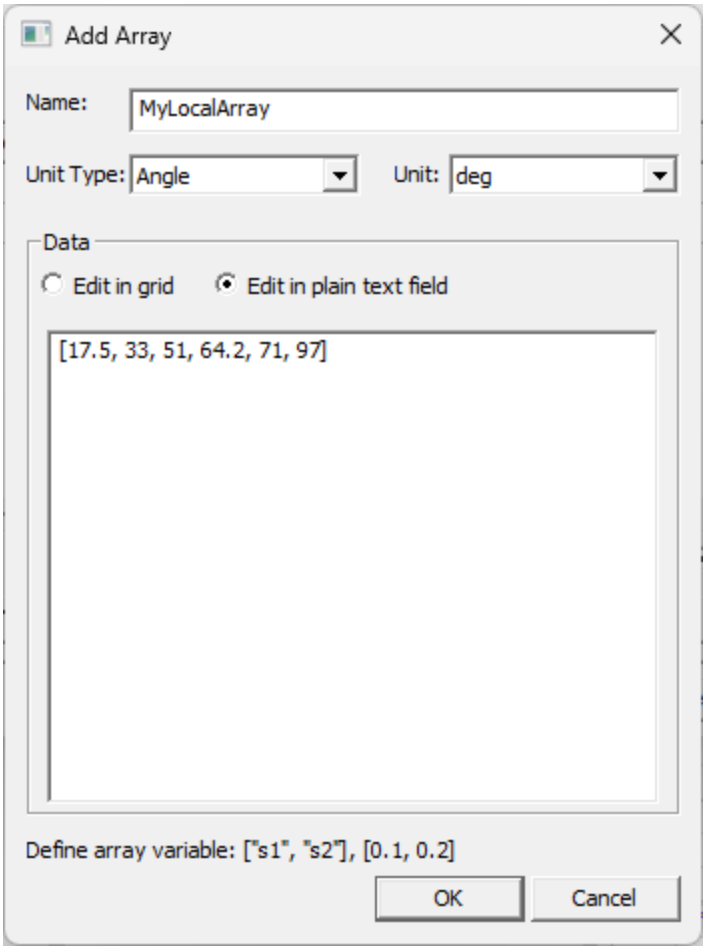
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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**.

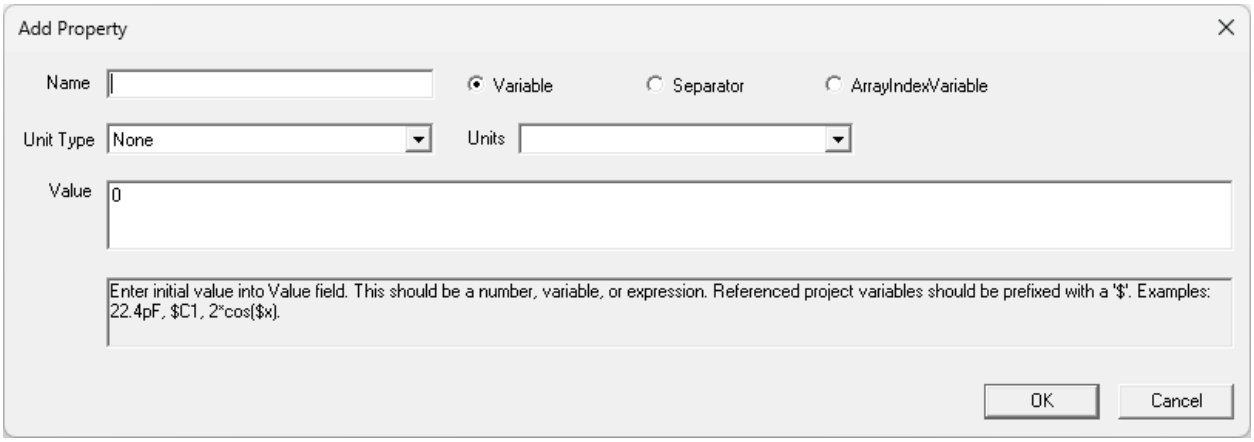
   The array appears in the **Local Variables** list.

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 **HFSS 3D Layout** > **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**.

Converting Variables and Parameter Defaults

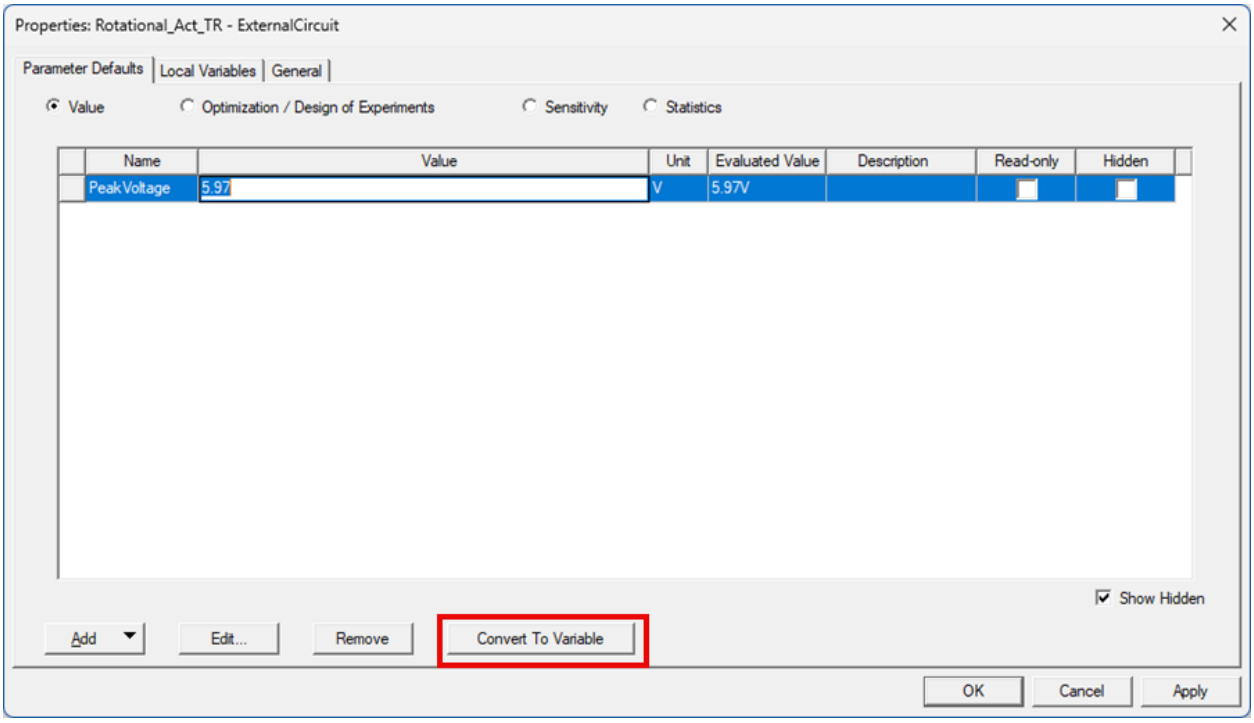
Design (Local) variables can be converted to parameter defaults, and vice versa.

To convert a parameter default or local variable:

1. Access **Design Properties** one of three ways:
   - Click **HFSS 3D Layout** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Design Variables**.

   The **Properties** window appears.
2. Select either the **Parameter Defaults** tab or the **Local Variables** tab, depending on which item you want to convert.

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 variable/parameter and click **Convert to Parameter** or **Convert to Variable**.

   The item will move to the list on the appropriate tab.

Some special cases to consider:

- If a local variable references another local variable, it cannot be converted to a parameter default. You will see a pop-up message if you attempt to convert. This is because local variables can reference parameter defaults, but parameter defaults cannot reference local variables.
- Parameter defaults can be overridden and can have different values in each subcircuit, while a Local Variable cannot. If a parameter default has been overridden and you convert it to a variable, it will lose all overrides. Attempting to do so will display a pop-up message asking if you really want to do the conversion and lose the override values.

Adding Variables and Parameter Defaults for Components

Circuit components and 3D layout components have parameter defaults and variables. Parameter defaults are encapsulated in the component. For an imported 3D layout component, the parameter defaults will be mapped as variables in the design the component is imported into.

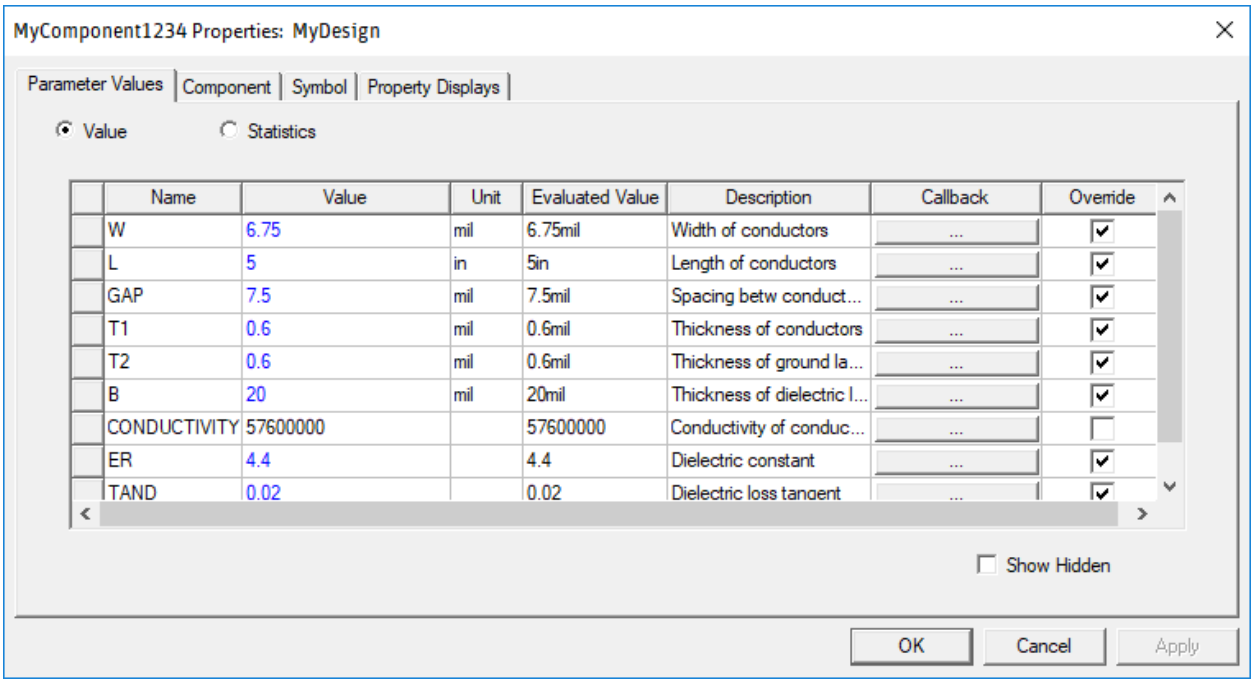
To define a new variable or parameter in a component within Circuit or HFSS 3D Layout:

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

1. Display a component's **Properties** window one of three ways:
   - Double-click the component.
   - Select the component and click **Edit** > **Properties**.
   - Select the component, right-click and select **Properties**.

   The **Properties** window appears.
2. Select the **Parameter Values** tab.

   The **Parameter Values** tab displays.



3. Select the **Value** radio button.
4. Select the **Value** field for the parameter you want to set equal to a local variable.
5. Type the variable name. The name may not be the same as an intrinsic variable or constant.

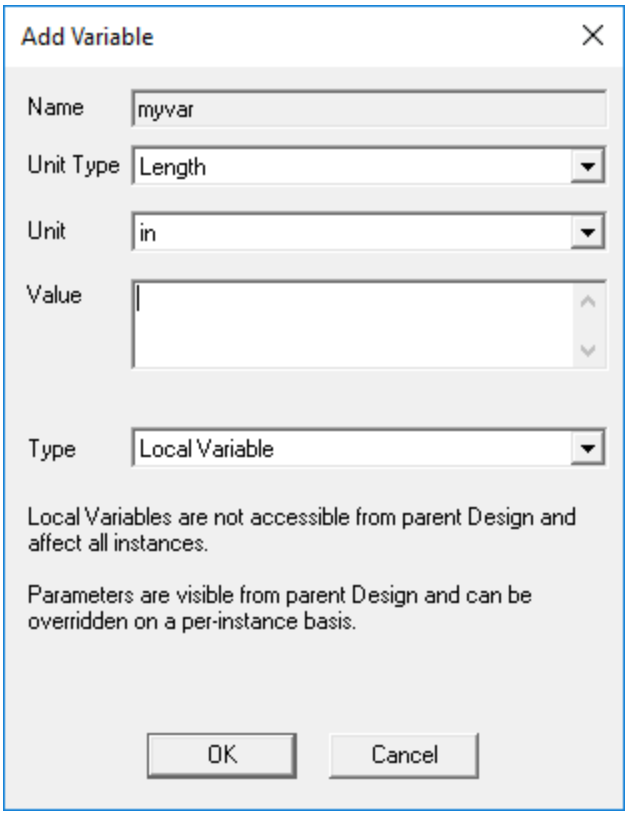
   Variable names may include alphanumeric characters and underscores (_).

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

> **Important:**
>
> If you wish to define a Project Variable, you must begin the name with a dollar sign ($).
>
> When you enter a name beginning with $, you will only be allowed to define it as a Project Variable.

6. Press **Enter**.

   The **Add Variable** window appears.



7. Select a **Unit Type** and **Unit** using the drop-down menus.
8. Enter a **Value**.

   The Value can be 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,

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

though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

> **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.

9. From the **Type** drop-down menu, select **Local Variable** or **Parameter Default**.
10. Click **OK** to exit the **Add Variable** window.
11. Click **Apply** to save your changes.
12. Click **OK** to exit the **Properties** window.

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

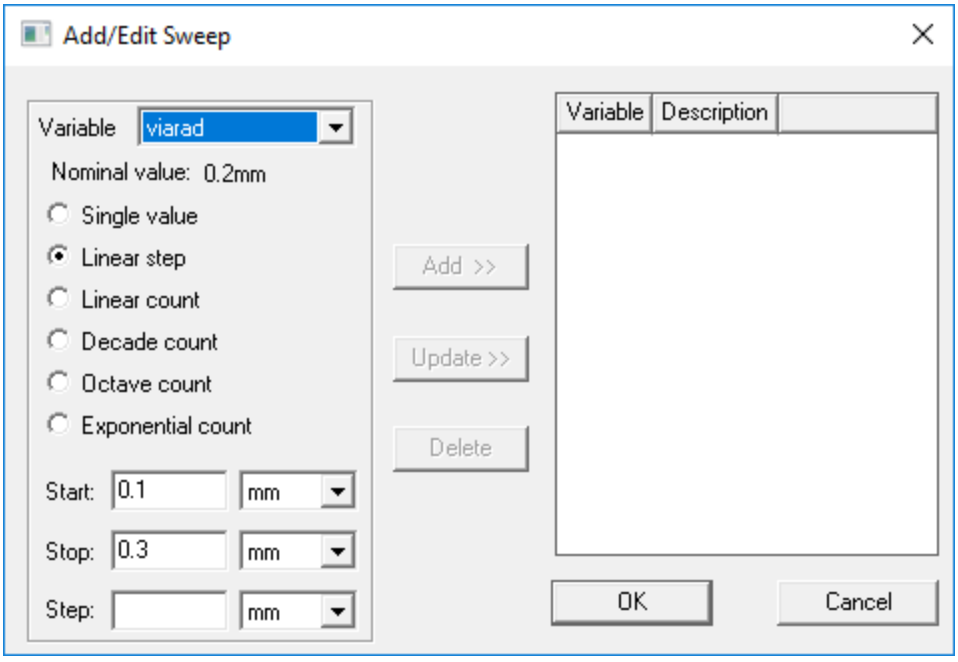
When setting up a sweep, the **Setup** window contains a list of variables being swept.

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

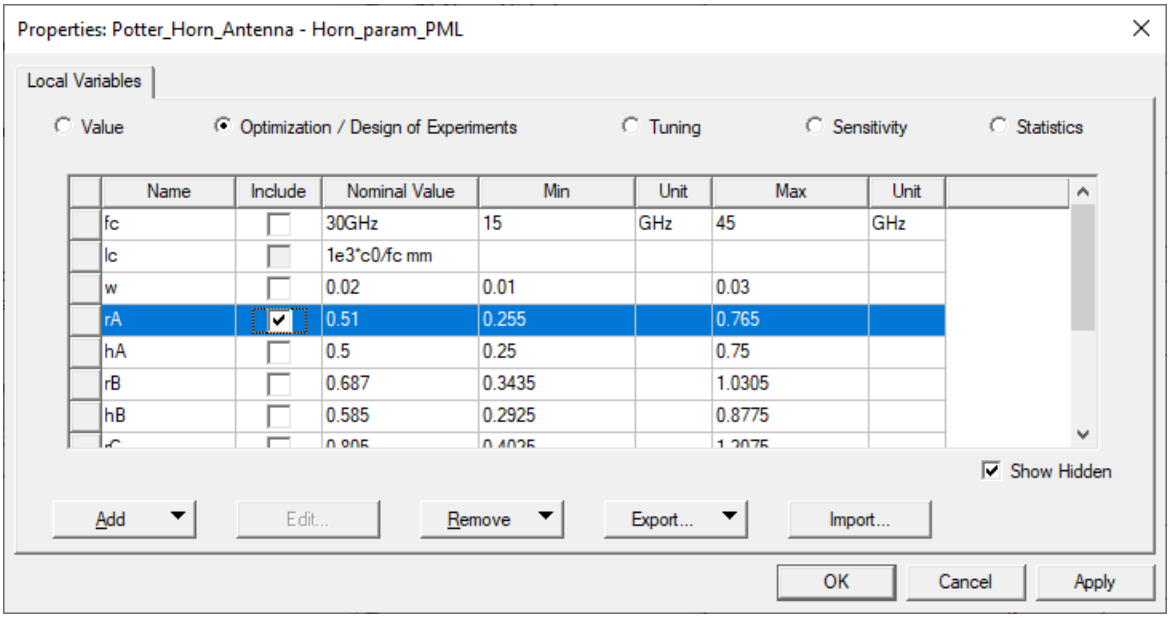
Before a variable can be optimized, you must specify that you intend for it to be used during an optimization analysis in the **Properties** window.

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 (**HFSS 3D Layout** > **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.

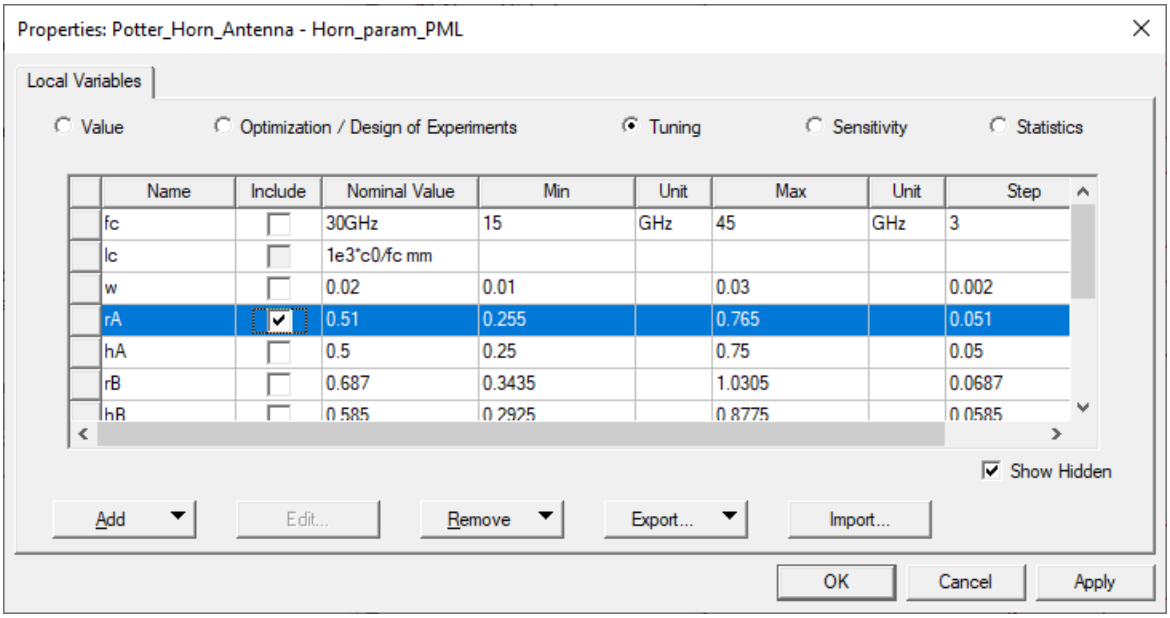
To specify that a variable is to be tuned:

1. Navigate either to the project variables **Properties** window (**Project** > **Project Variables**), or to the design variables **Properties** window (**HFSS 3D Layout** > **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

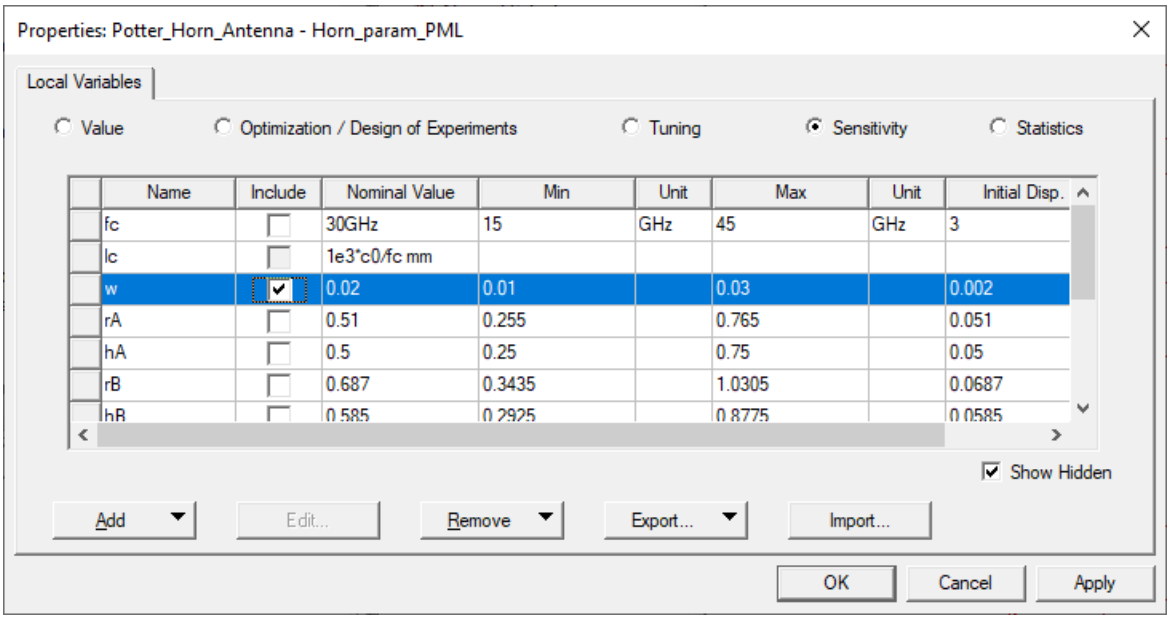
Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Properties** window.

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 (**HFSS 3D Layout** > **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:

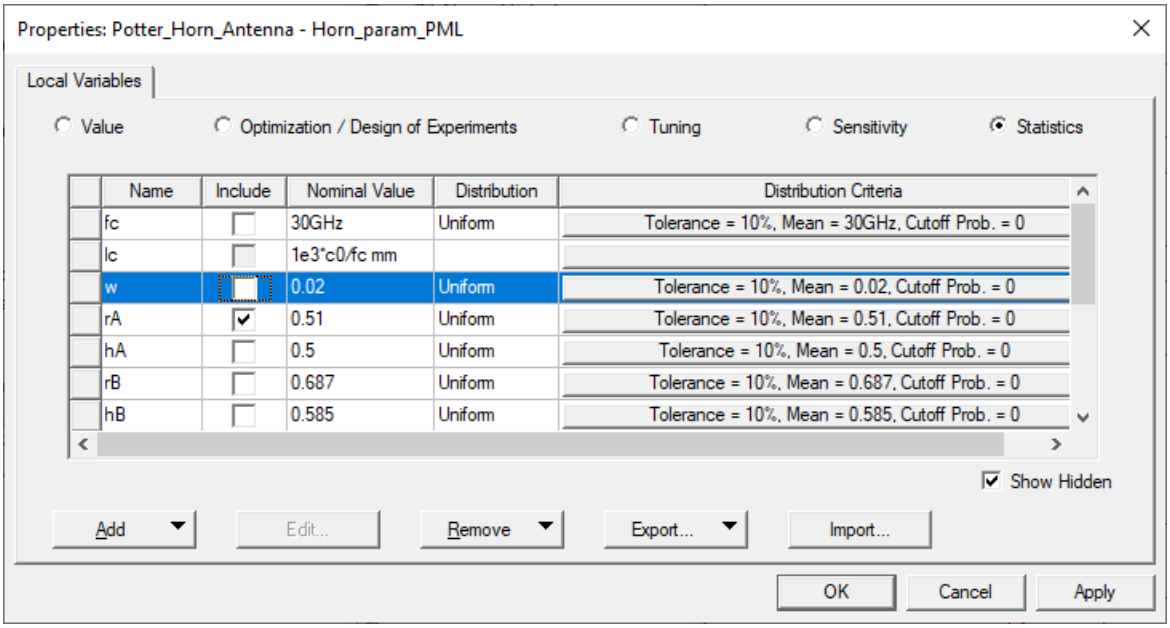
1. Navigate either to the project variables **Properties** window (**Project** > **Project Variables**), or to the design variables **Properties** window (**HFSS 3D Layout** > **Design Properties**).

   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.

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 **HFSS 3D Layout** > **Design Datasets**.

Adding Datasets

The following procedure describes how to add a dataset manually.

You can also add datasets:

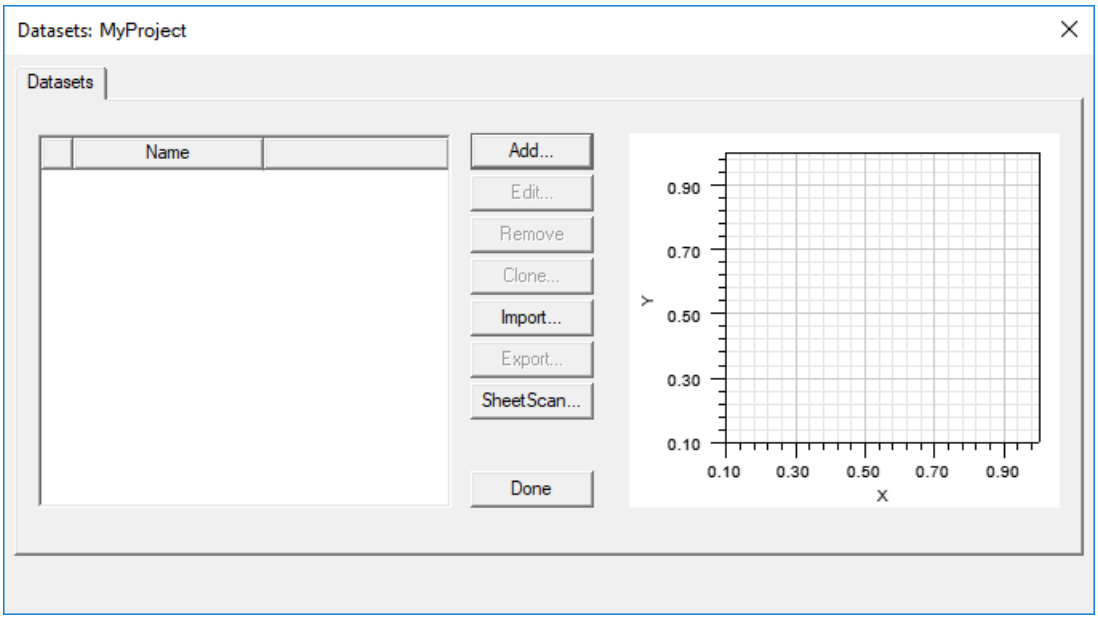
- By importing data from an external tab-delimited file using the **Import...** button.
- Using the Sheetscan tool.

To add a dataset manually:

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** from the menu bar, or click **Datasets** on the **Desktop** ribbon tab.
   - For design-level datasets, click **HFSS 3D Layout** > **Design Datasets** from the menu bar.

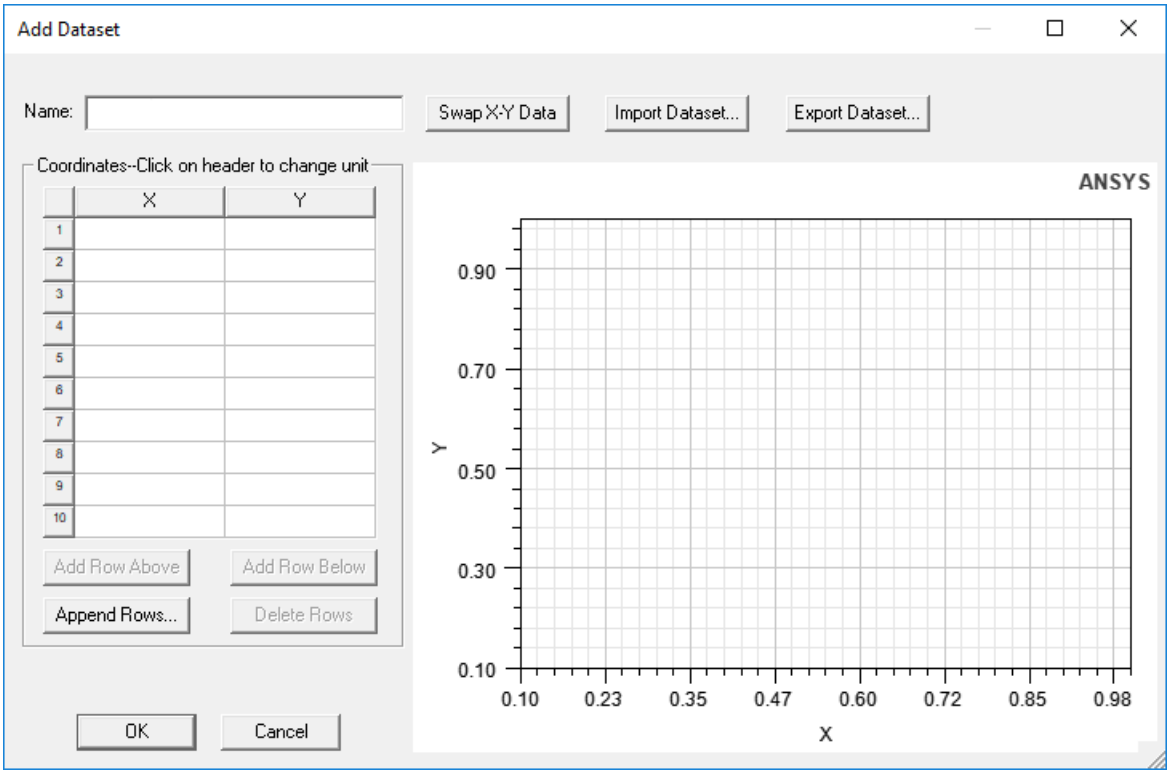
   The **Datasets** window appears, listing any project or design-level datasets.



2. Click **Add**.

   The **Add Dataset** window appears.

Ansys Electromagnetics Suite 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.

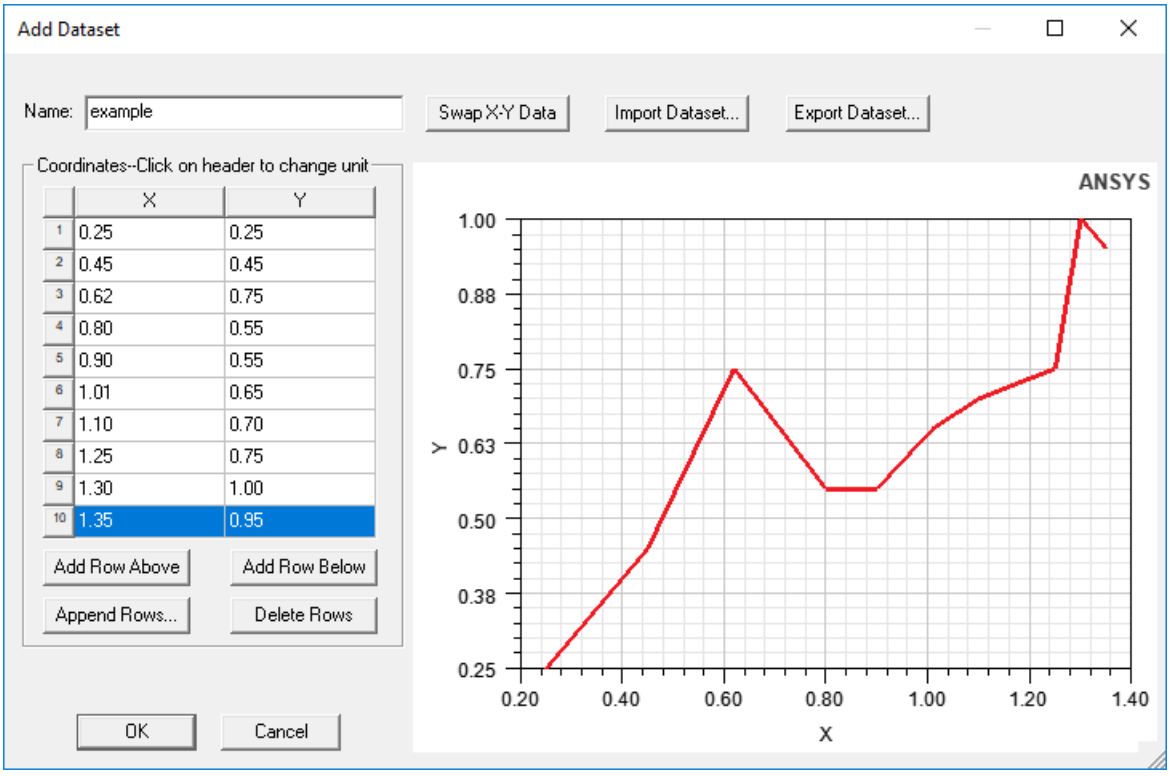
3. Enter a **Name** for the dataset.

> **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.

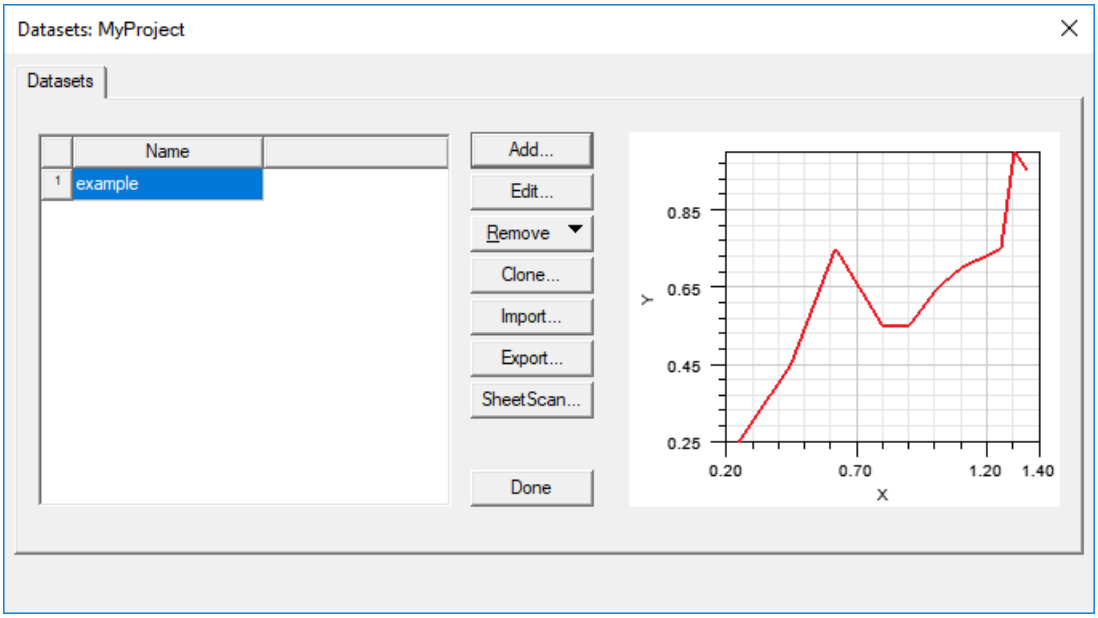
   The display updates to show your coordinates.



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.

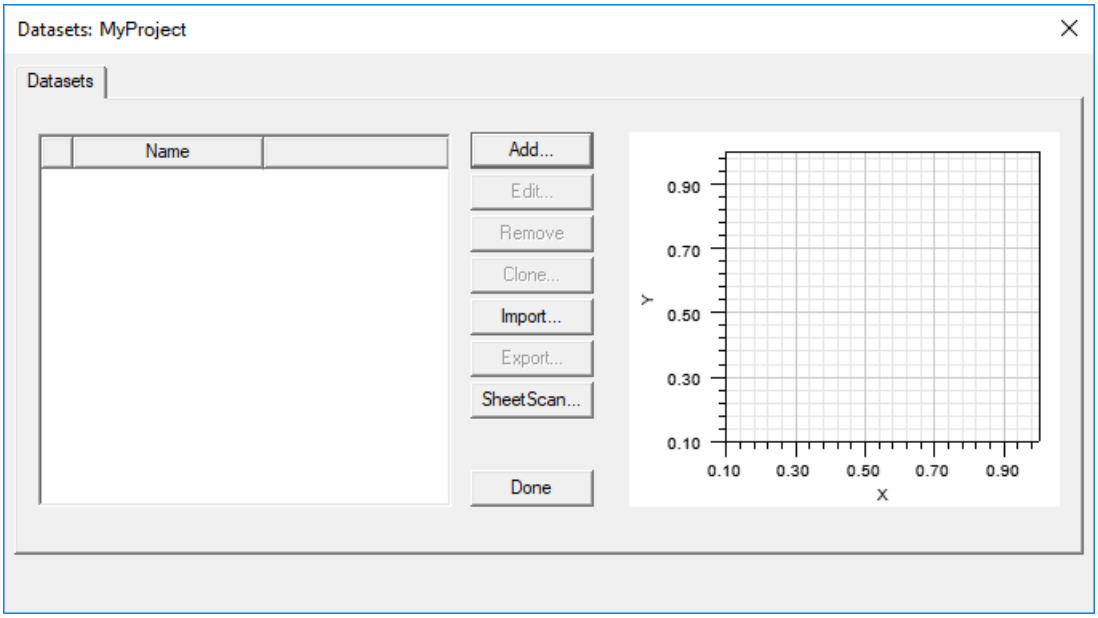
Importing Datasets

To import a dataset from a file:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS 3D Layout** > **Design Datasets**.

   The **Datasets** 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 **Import**.
3. In the file browser that appears, use the drop-down menu to select the file format of the file you wish to import.

> **Note:**
>
> The table below lists the file types supported for *Project level datasets*. For *Design level datasets*, only **.tab** files are supported.

.mdx, .mda	Twin Builder Characteristic format
.xls, .xlsx	Microsoft Excel
.txt	text file
.csv	Comma-separated value
.out	Maxwell SPICE (read-only - reads data inside the KW_ DATA section)
.cfg	Comtrade (IEEE Std C37-111-1999)
.dat	TEK Oscilloscope
.tab	tab delimited data files

   a. If you select an **.mdx**, **.mda**, **.out**, **.cfg**, or **.tab** file, the data is imported immediately into **Datasets** window.
   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.

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

> **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.

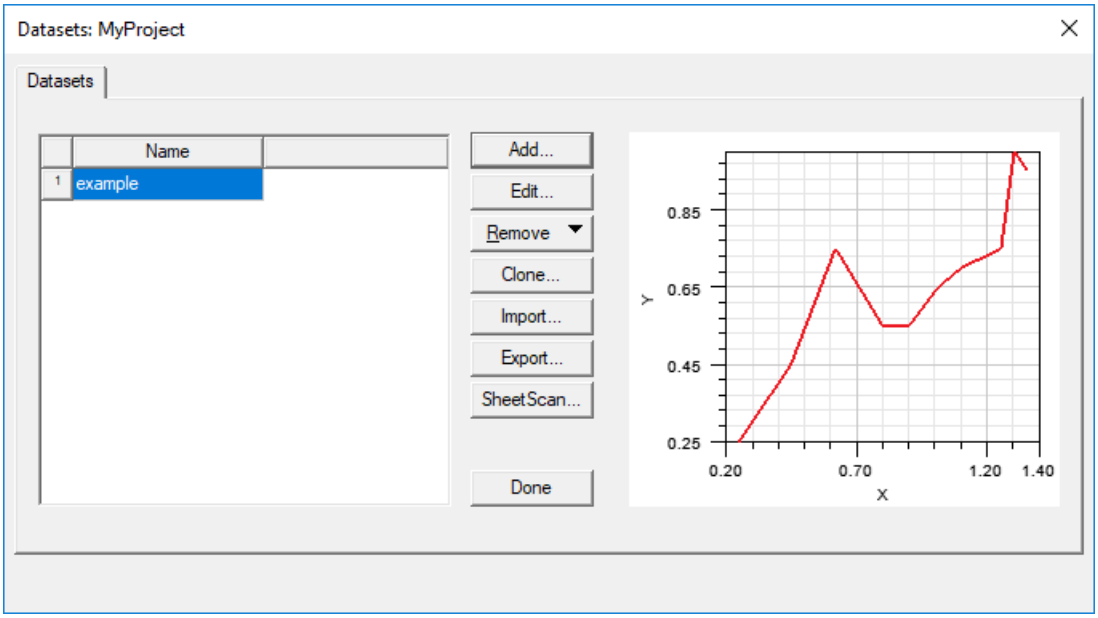
Editing Datasets

To modify an existing dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS 3D Layout** > **Design Datasets**.

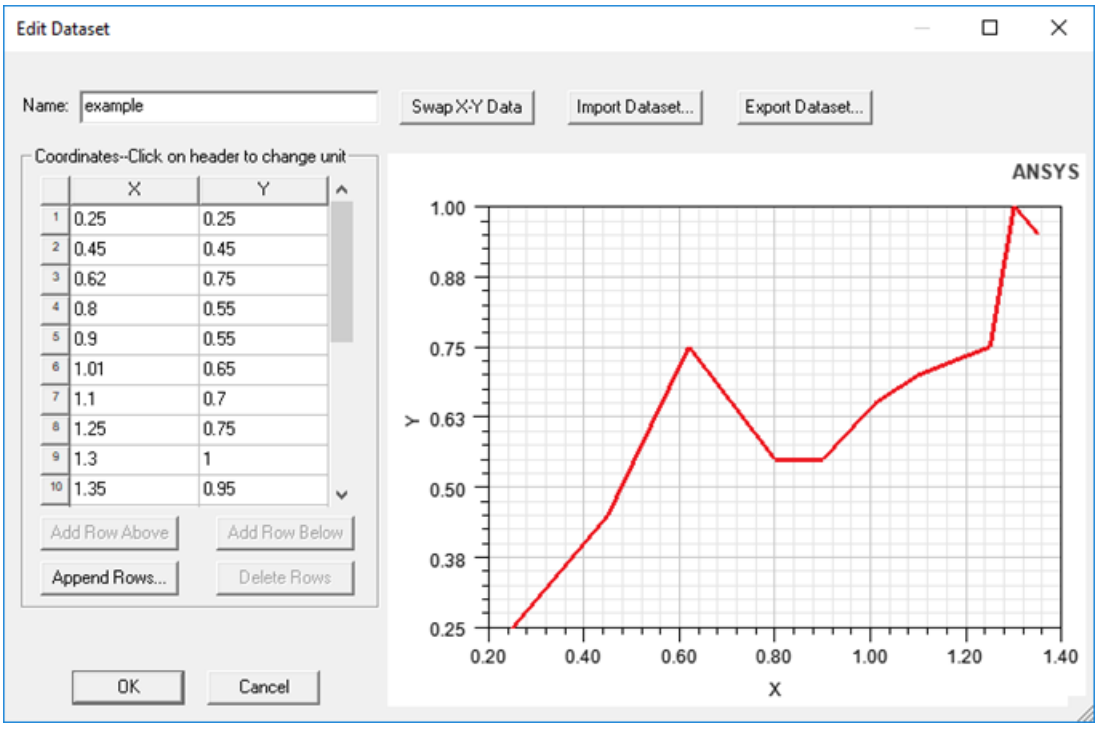
   The **Datasets** window appears, listing any project or design-level datasets.

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.

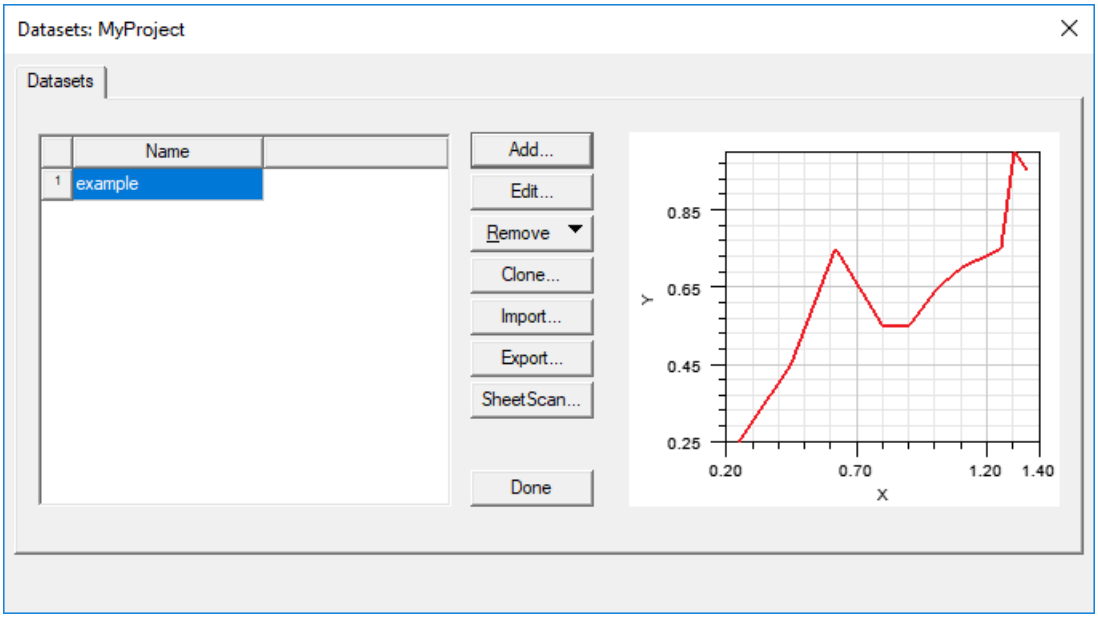
Cloning Datasets

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 **HFSS 3D Layout** > **Design Datasets**.

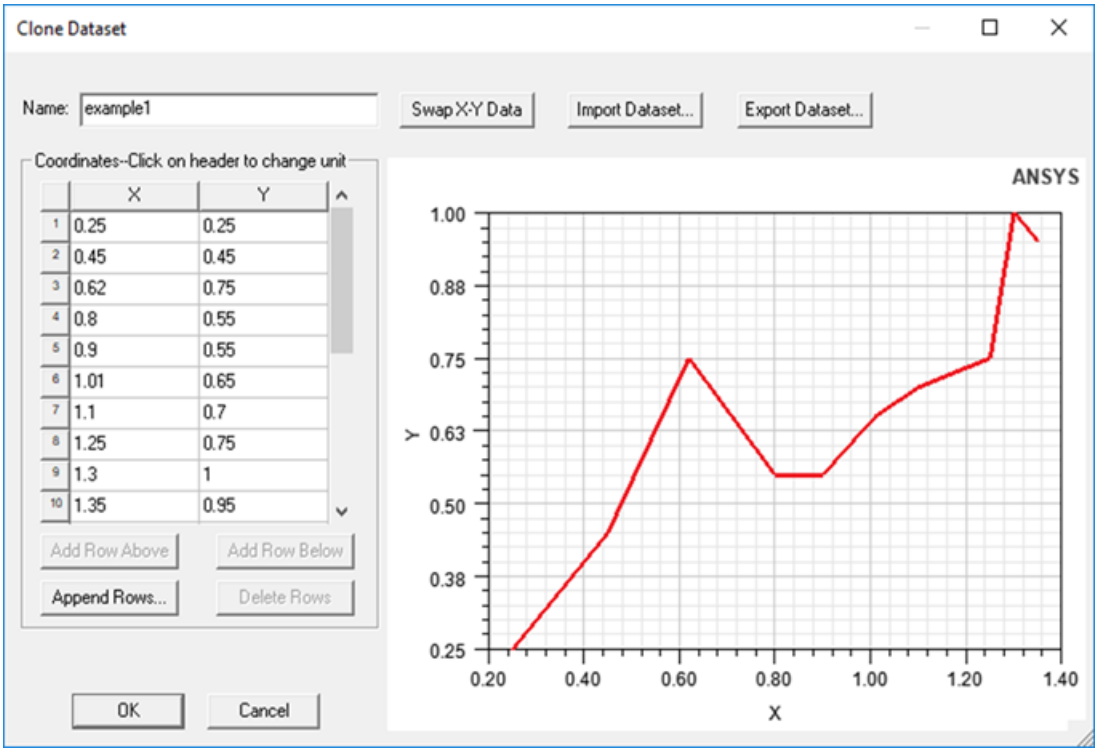
   The **Datasets** window appears, listing any project or design-level datasets.



2. Click **Clone**.

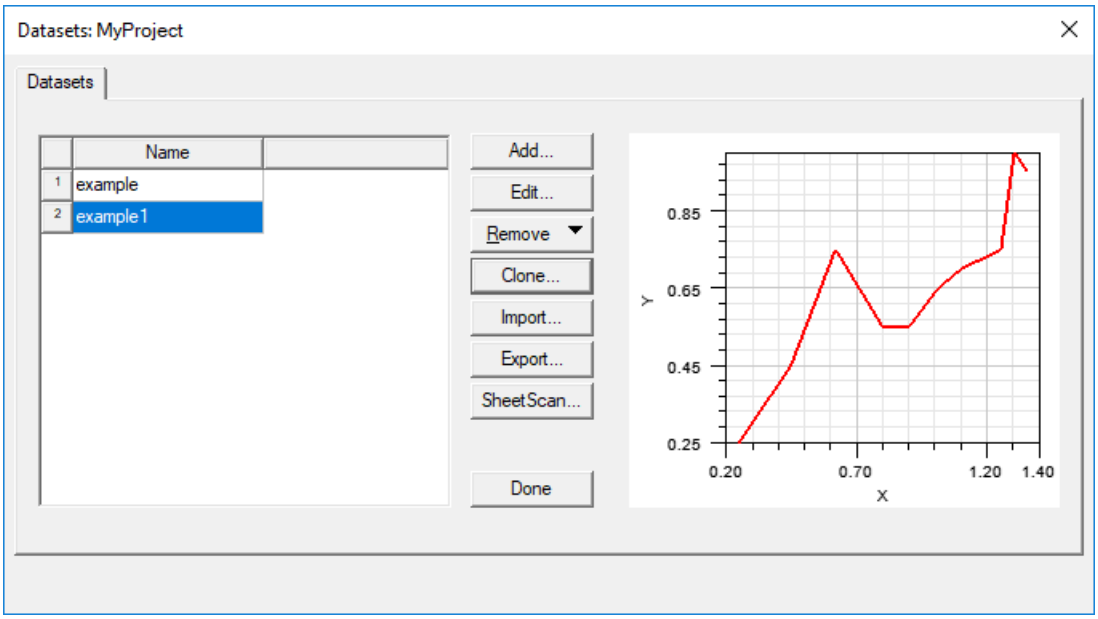
   The **Clone Dataset** window appears. The default name is the original dataset name with a number appended to the end.

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.

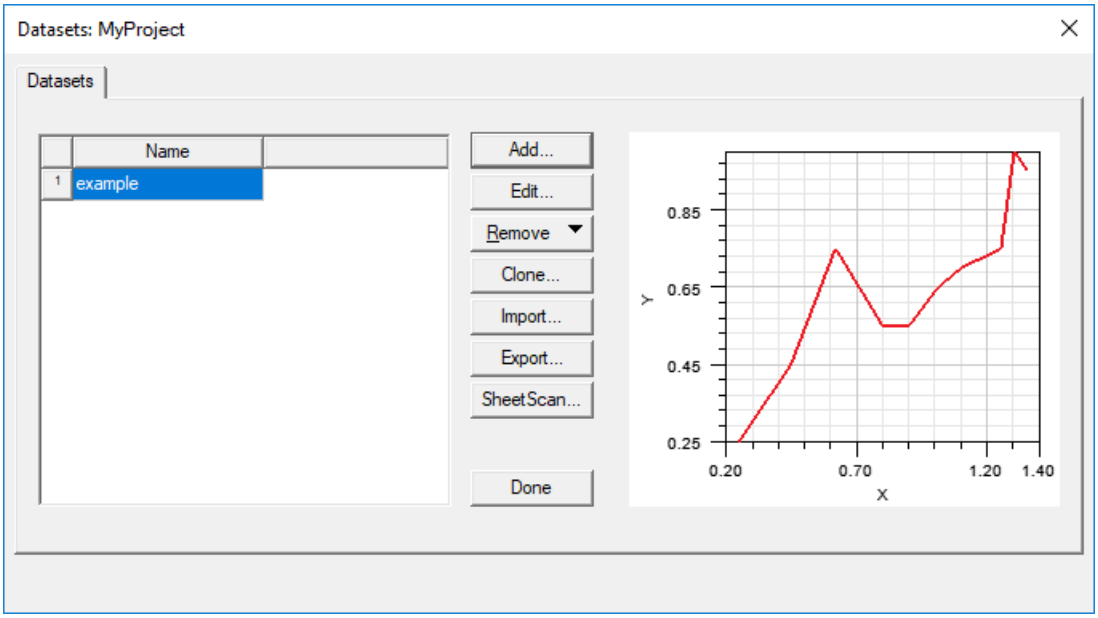
Exporting Datasets

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 **HFSS 3D Layout** > **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.

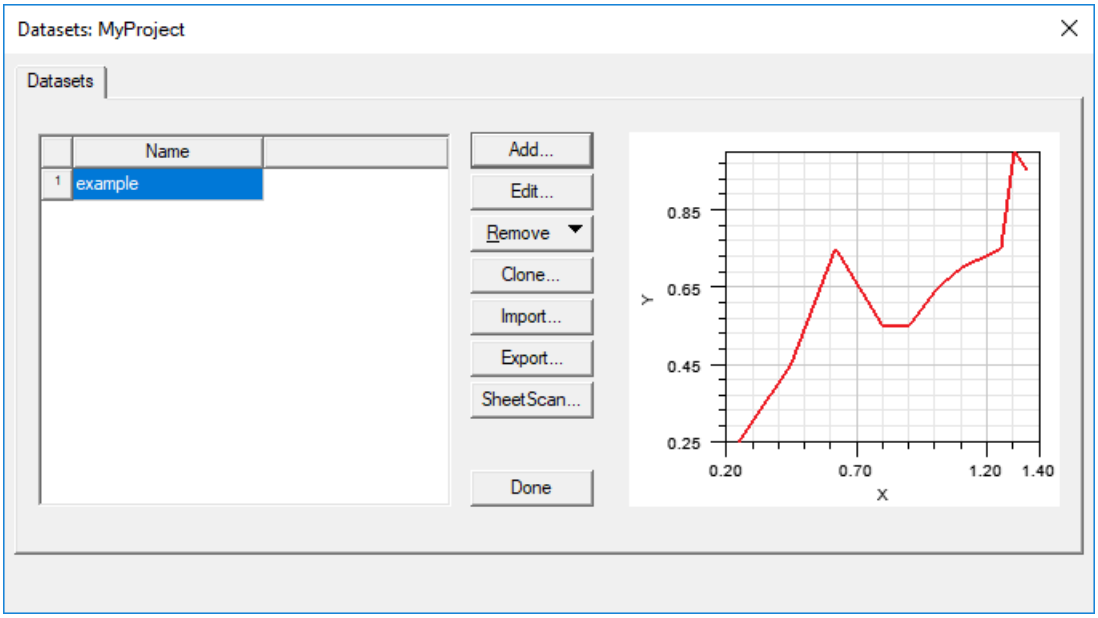
Removing Datasets

To remove a previously added dataset:

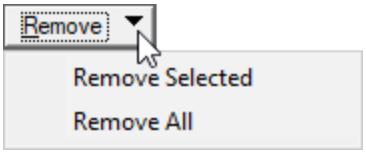
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS 3D Layout** > **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.

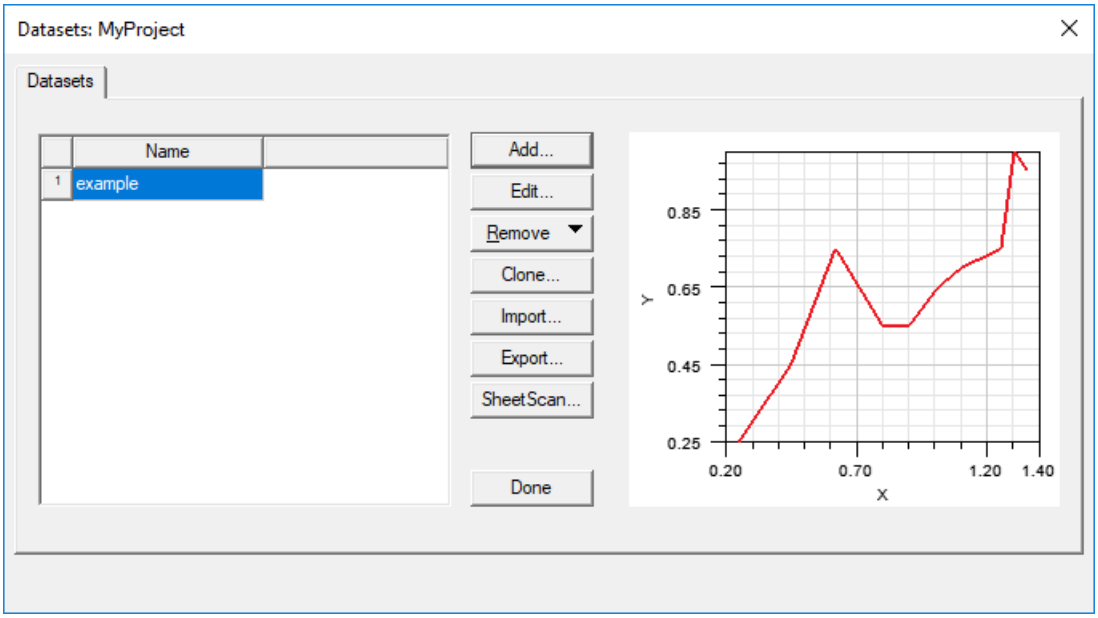
> **Note:**
>
> The **Remove All** option does not work across dataset types. That is, selecting **Remove All** from the **Project Datasets** window does not remove design-level datasets, and vice versa.

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

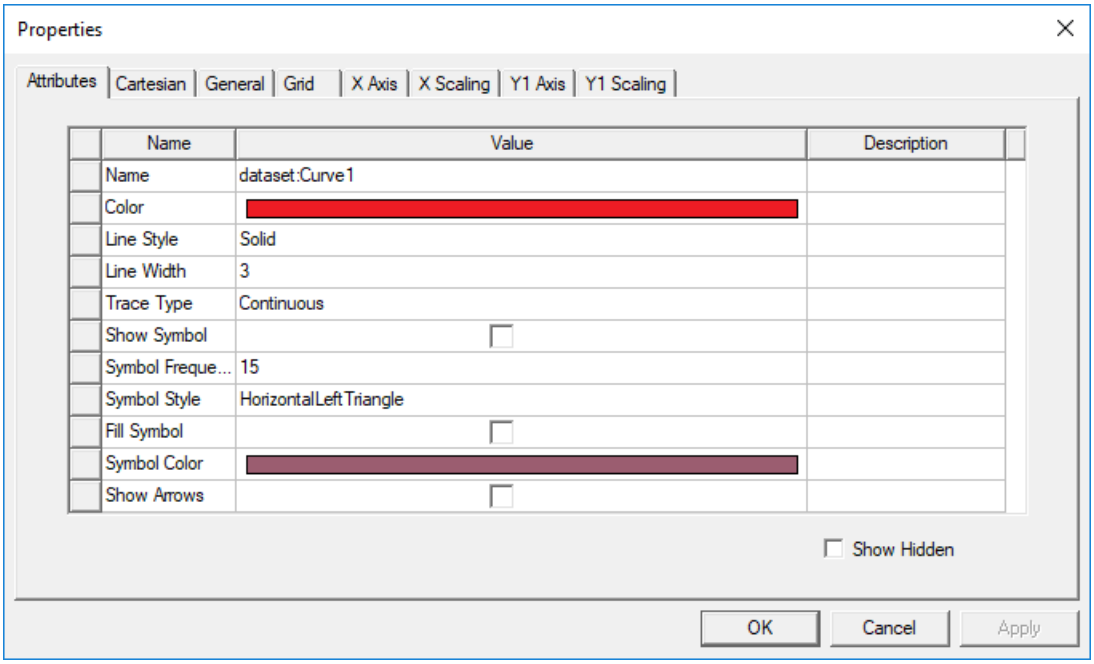
From the **Datasets** window, you can change the appearance of a dataset's plot.



To change a plot's appearance:

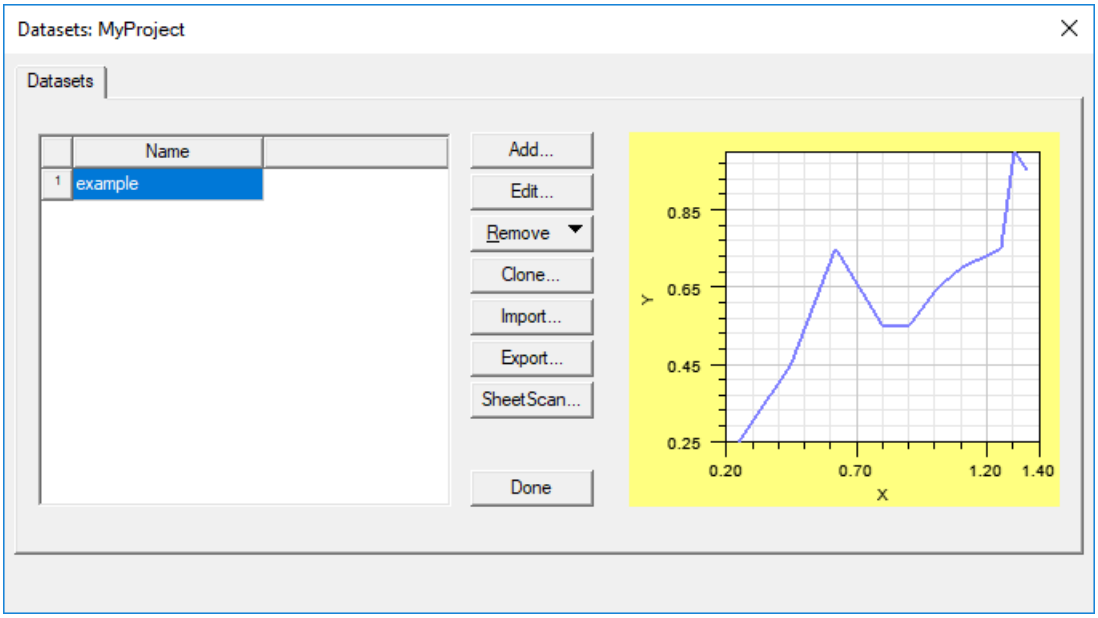
1. Double-click a plot element (for example, a line, axis, or title) to open that element's **Properties** window.

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 **HFSS 3D Layout** > **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.

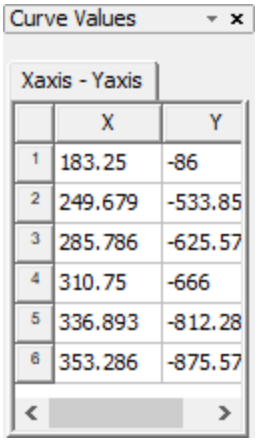
- **Curve** – contains a drop-down menu allowing you to select a curve and tools for working with curve values.

- **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.

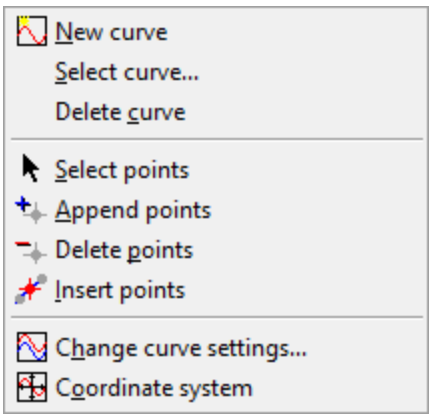


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

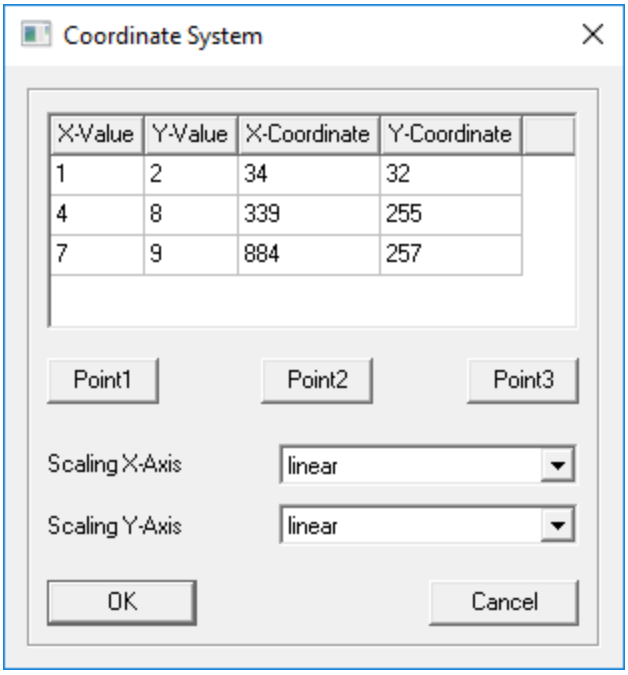
Defining a SheetScan Coordinate System

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**.

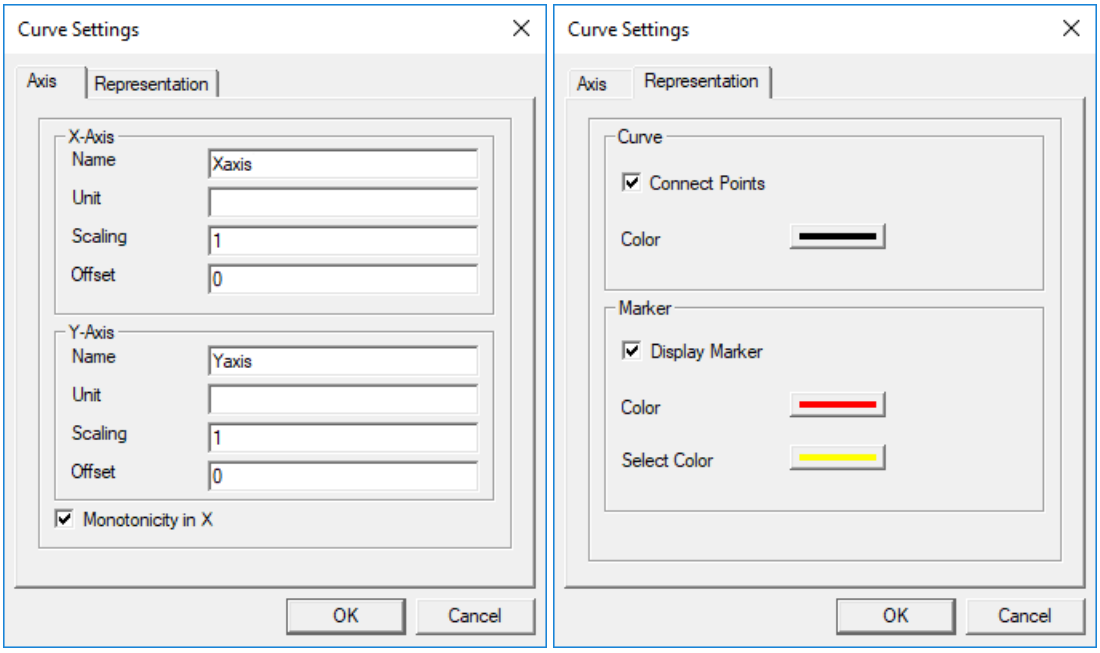
8. You can now define a Characteristic Curve.

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.

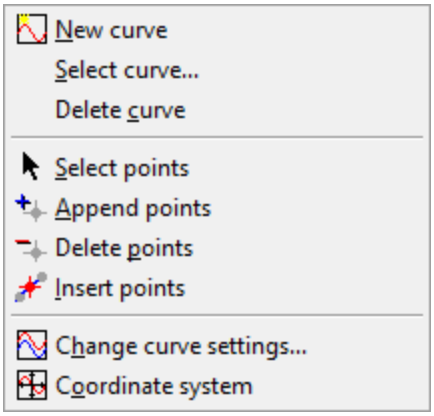
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

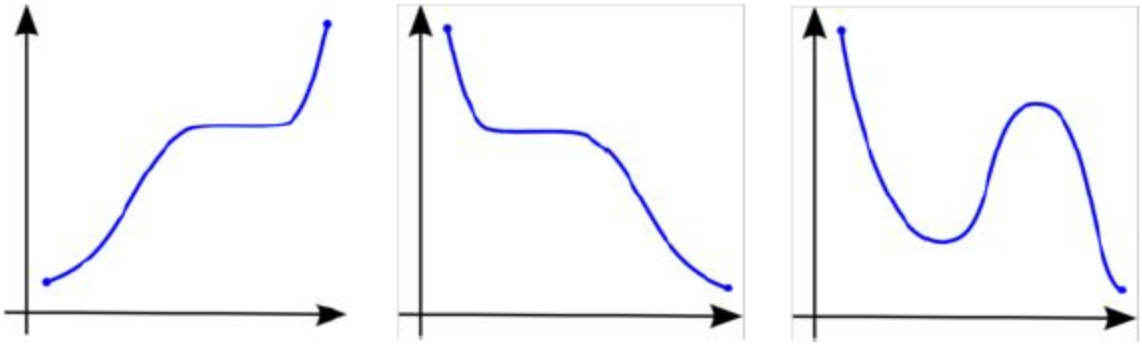
Checking for Monotonicity in X

HFSS 3D Layout 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.

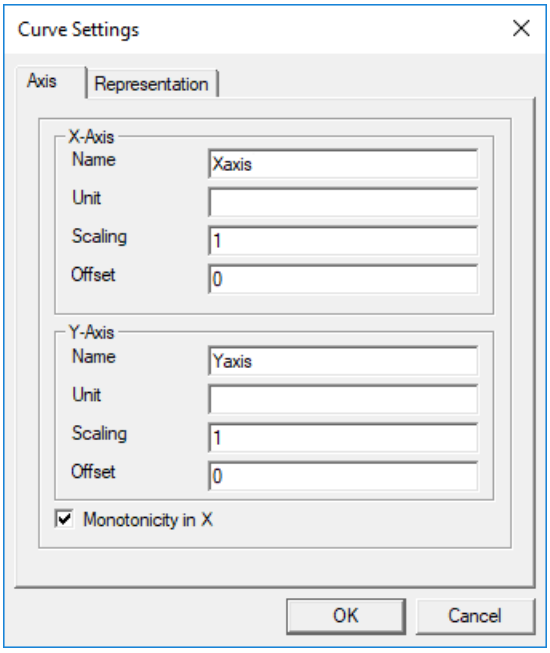
      > **Note:**
      >
      > - Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
      > - The first 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:

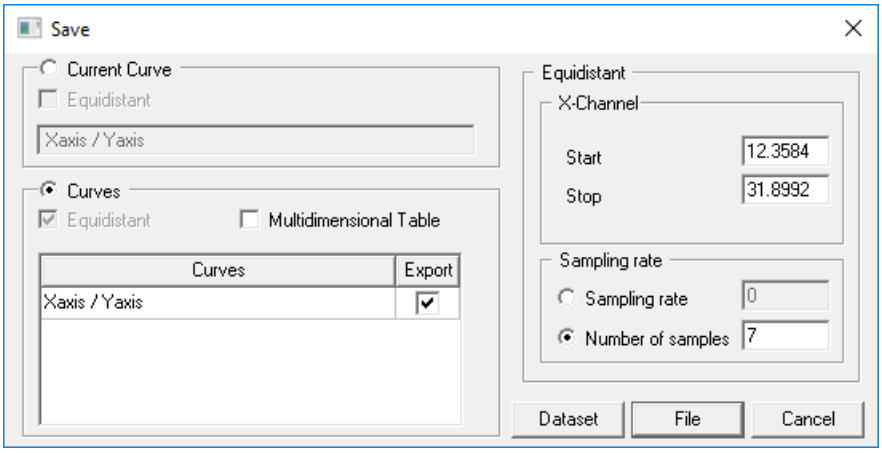
- Electronics Desktop Dataset
- Twin Builder Characteristic (*.mdx) file
- Comma Separated Value (*.csv) file
- Comtrade (*.cfg) file

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**.

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.

Valid Operators for Expressions

The operators that can be used to define an expression or function are performed in a sequence.

The following table lists valid operators and the sequence in which they are accepted (listed in decreasing precedence):

()	Parenthesis	1
!	Not	2
^ (or **)	Exponentiation (If you use "**" for exponentiation, as in previous software versions, it is automatically changed to "^".)	3
-	Unary minus	4
*****	Multiplication	5

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

/	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

HFSS 3D Layout 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
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)

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

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 (e^x)	exp(x)
pow	Raise to power (x^y)	pow(x,y)
if	If	if(cond_exp,true_exp,false_exp)
pwl	Piecewise Linear. (**pwl** can be used with datasets for Design Variables but not for Project variables).	pwl(dataset_exp,variable)
pwl_periodic	Piecewise Linear for periodic extrapolation on x.	pwl_periodic(dataset_exp,variable)
sqrt	Square Root	sqrt(x)
ln	Natural Logarithm (The "log" function has been discontinued. If you use "log(x)" in an expression, the software automatically changes it to "ln(x)".)	ln(x)
log10	Logarithm base 10	log10(x)
int	Truncated integer function	int(x)
nint	Nearest integer	nint(x)
max	Maximum value of two parameters	max(x,y)
min	Minimum value of two parameters	min(x,y)
mod	Modulus	mod(x,y)
rem	Returns the fractional part of a decimal number such that rem(x) = x-int(x)	rem(x)
clp	Implements smooth interpolation employing weighted impact of all points of the dataset (not just the closest one). See formula below. **Note:** If used with a large 3D dataset, clp function will degrade.	clp(*datasetName*, X,Y,Z)

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

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.

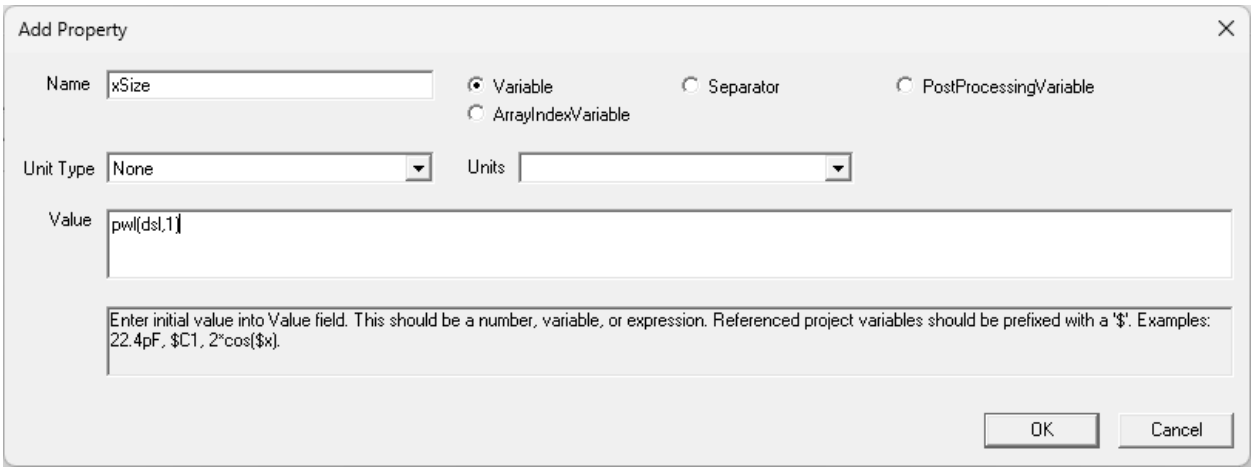
You can use **pwl** in an expression that uses datasets for such things as a frequency dependent material property (See: Adding Datasets).

For example, you can specify BulkConductivity as:

```
pwlx($ds1, Freq)
```

You can create a design variable representing a dimension `xSize` as `pwl(ds1, 1)` where `ds1` is a dataset.

Doing so looks like this:

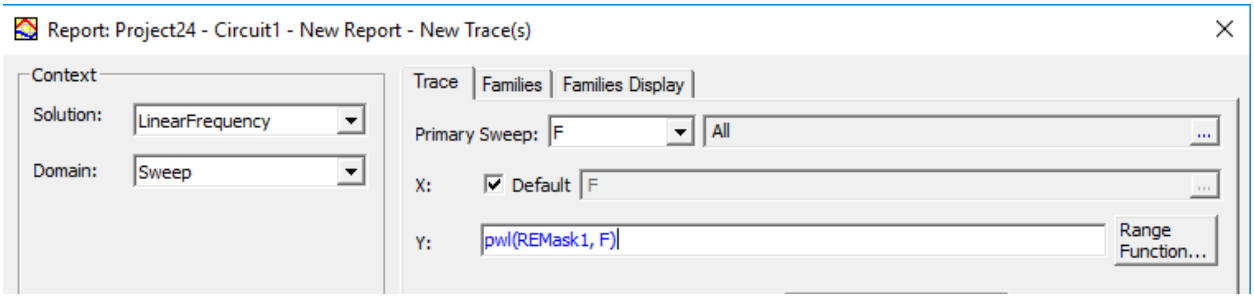


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

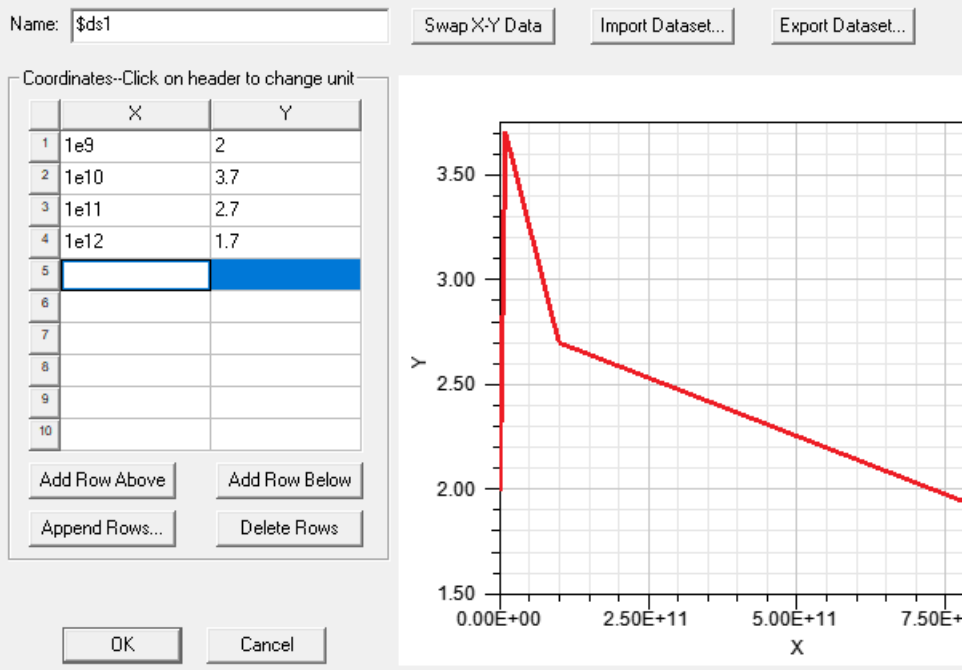
> **Note:**
>
> The following is an example showing how to make a material property frequency dependent using a dataset as the first parameter to a **pwl** function. The values used are arbitrary.

1. For a project-level dataset, click **Project** > **Datasets**. For a design-level dataset, click **HFSS 3D Layout** > **Design Datasets**.

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

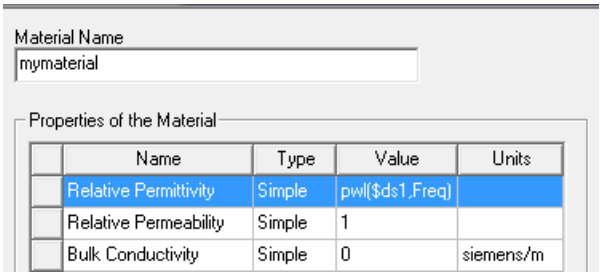
2. Click **Add**.
3. Set the **Name** field and **Coordinates** as desired and click **OK**.

   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.



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 **OK**.

Cosimulation of Circuit and Field Solver Designs

The Electronics Desktop features Dynamic Links and Solver on Demand™ technology to allow flexible and accurate modeling of unrestricted structures used in high-frequency system designs of components and circuits. Dynamic Links allows the Circuit project to incorporate projects from HFSS, Q3D, 2D Extractor, and SIwave as active components. Use Solver on Demand technology to simulate structures via circuit, system, and field solver models at the touch of a button. Solver on Demand incorporates electromagnetic solver results directly into larger network topologies and allows you to maximize both speed and accuracy. Solution data is cached and used in place of resimulating unless you change structure parameters.

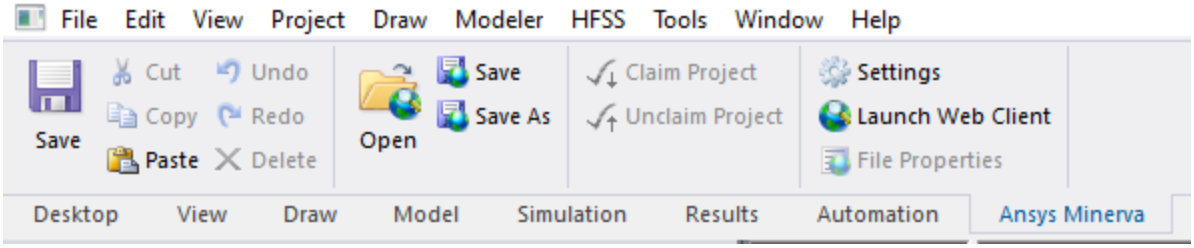
For more information see Dynamic Links and Solver On Demand.

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.

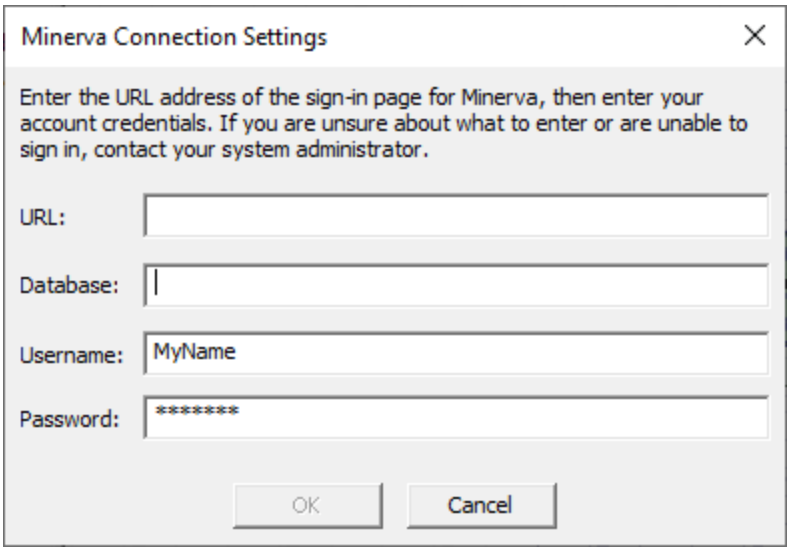
4 - Minerva Remote Storage Environment

Ansys Minerva is a remote storage environment where you can store Ansys project archive files. You can collaborate with other users by downloading a project, making changes, and then uploading the project. One of the features of Ansys Minerva is the ability to store and retrieve Ansys projects from the Minerva server. This ability has been integrated into Electronics Desktop so it is now possible to download and open a project stored on Minerva directly from the Electronics Desktop interface. In the same way, it is also possible to save a project to Minerva. The image below shows the Minerva ribbon containing Minerva commands accessible through the desktop.



Minerva has its own HTML web interface with additional features as a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos.

Minerva Settings

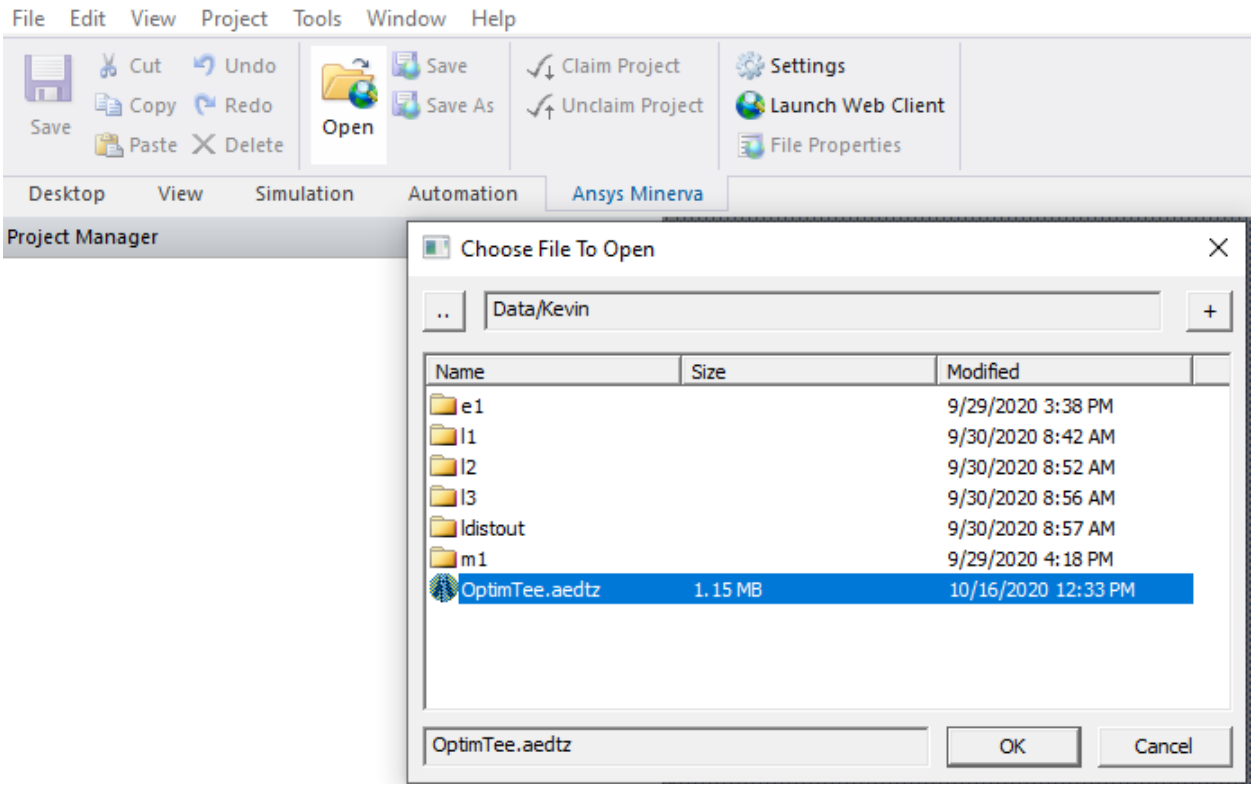


When you first login via the desktop, or to the Web Client, you must provide a Minerva URL, a database name, a user name, and a password. You will need to obtain these from your system

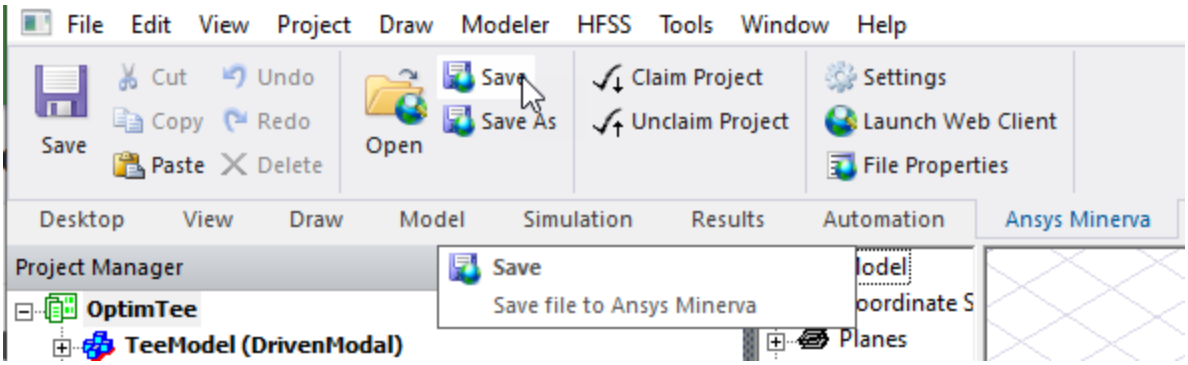
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

administrator. Once you provide the connection settings, the other Minvera commands are enabled.

Open – opens the Choose File to Open browser that lets you select and download an archive file, extract it, and open it. The [..] button moves up one level in the file hierarchy. The [+] button creates a new folder at the current level.

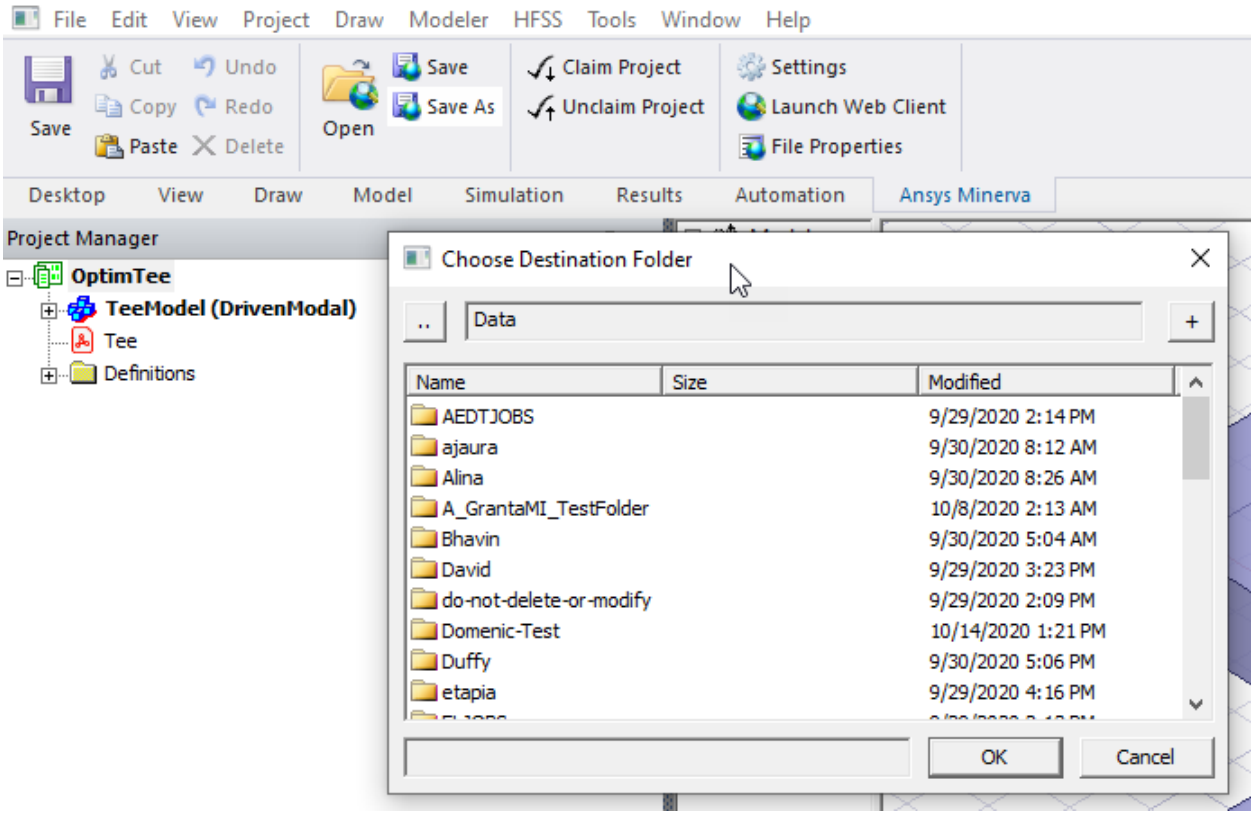


Save – archives the current project, and saves it to a remote Minerva system.



Save As – prompt for remote file, archive project, save to remote Minerva system.

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



In order to prevent users from overwriting each other's work, Minerva has added the concept of claiming and unclaiming files. A file claimed by one user cannot be overwritten on the Minerva server by another user. Unclaimed files can be overwritten by anyone at any time. Once a file has been claimed by one user, it cannot be claimed by a different user until it has been unclaimed by the original claimer. Note: even if a project is claimed by one user, other users are free to download, open and edit the file locally.

The recommended flow when working on a project is

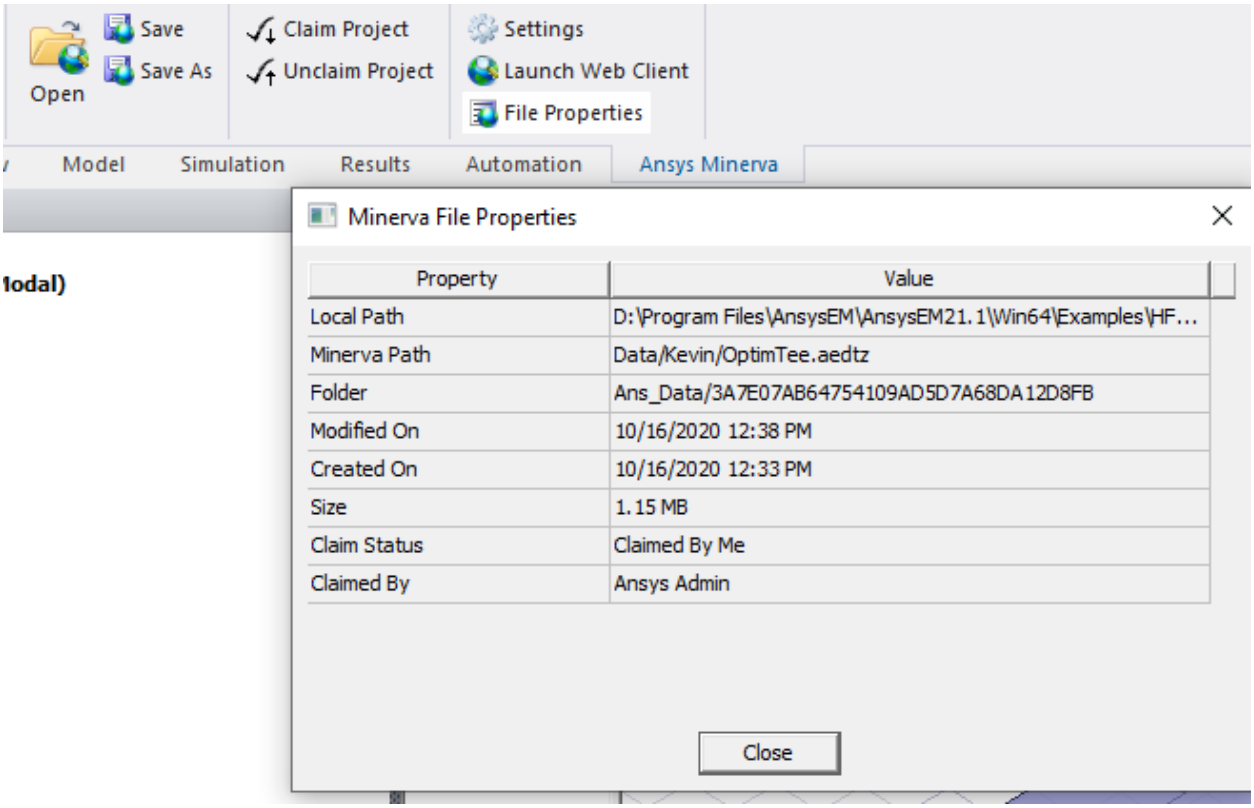
1. Claim the file, make sure you are working with the most recent version
2. Make edits
3. Save the file to Minerva
4. Unclaim the file

Claim Project – claims file so it cannot be modified by other users.

Unclaim Project – releases claimed file.

File properties – opens a dialog letting you view remote file properties. Notice the Claim Status and Claimed By properties.

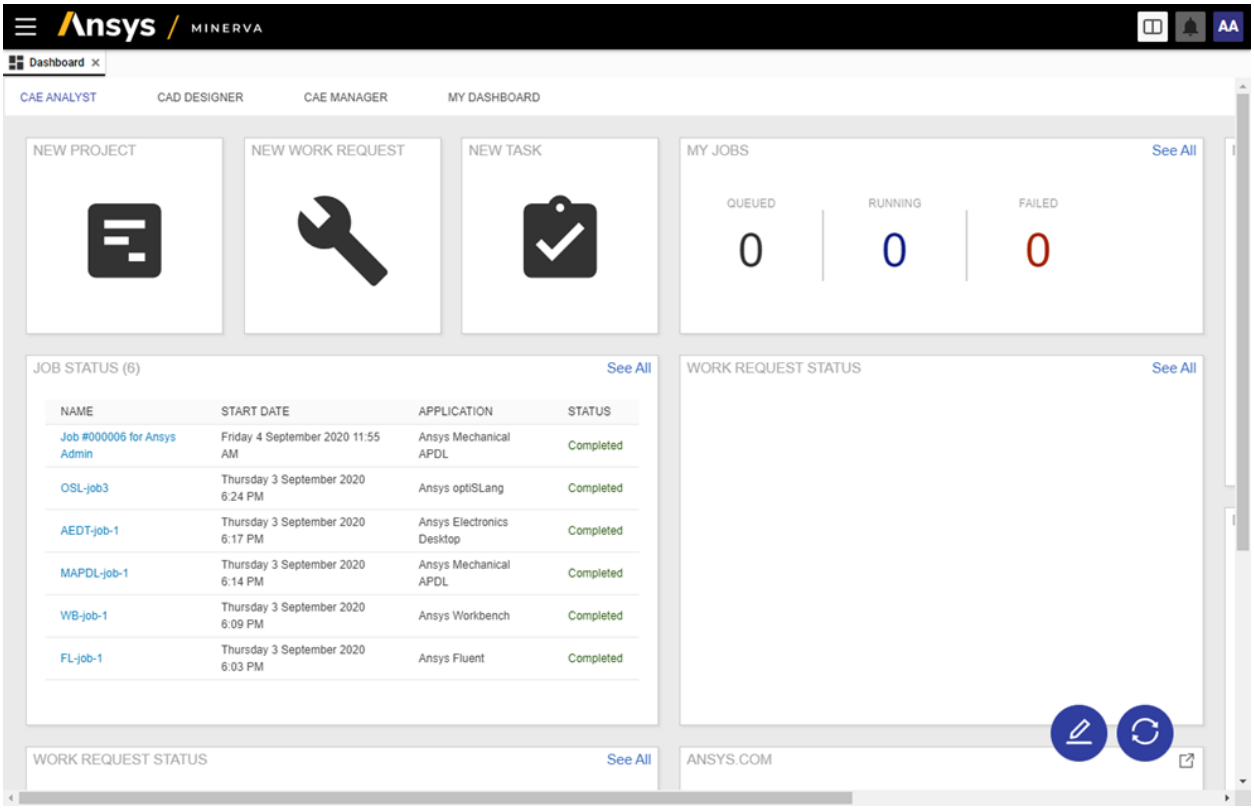
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Launch Web Client:

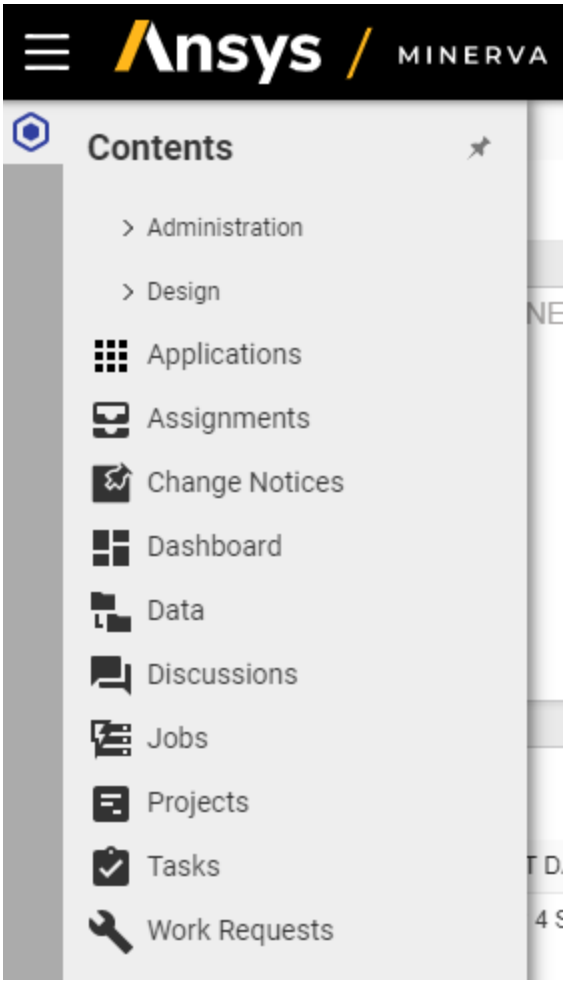
Ansys Minerva, powered by Aras, is a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos. Launching the Minerva Web Client gives access to these features. This opens a dialog calling for you to specify provide login information. It attempts to open a web browser window, so your default browser must be compliant.

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



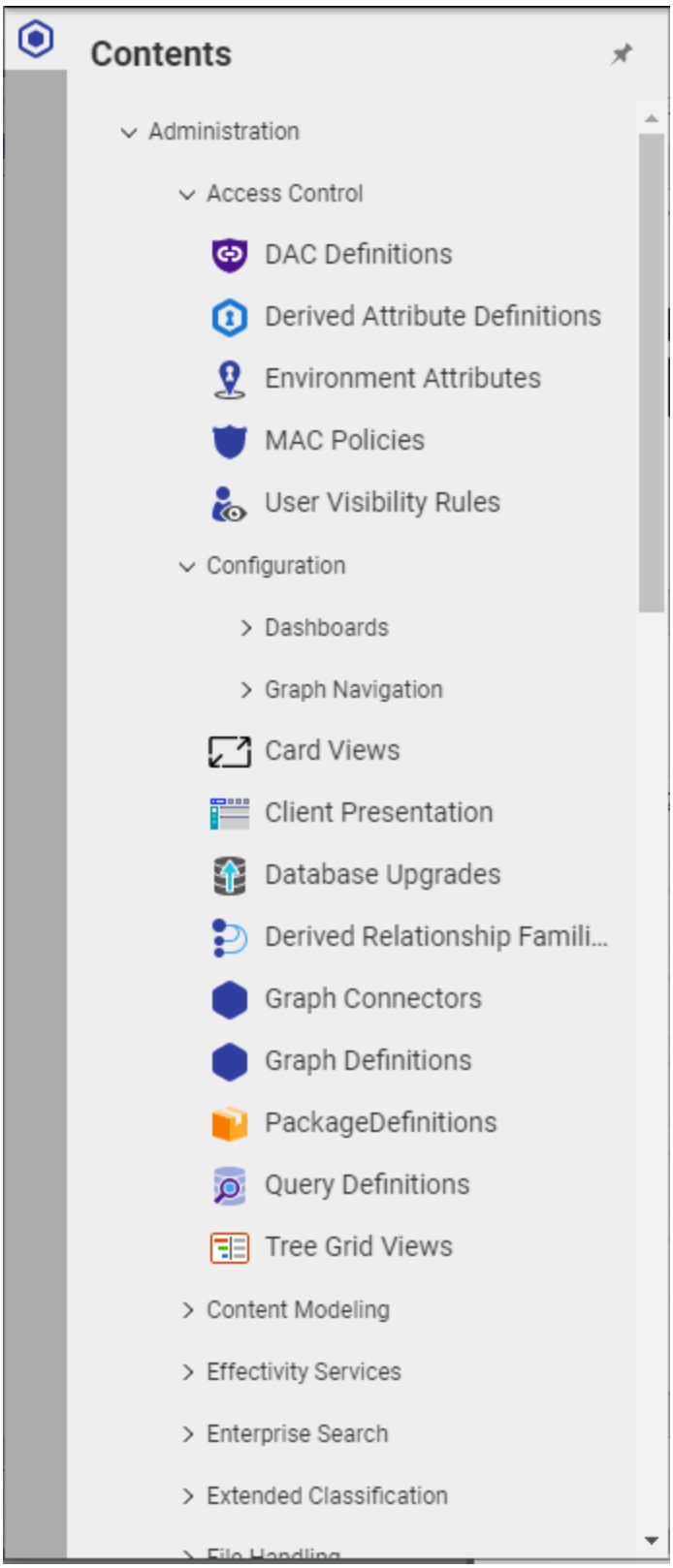
Clicking on the triple bar icon left of the Ansys logo opens up access to the more robust Minerva Web Client functionality.

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



The arrow icons for Administration open up further functionality.

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.

5 - Layout and Circuit Import Export Operations

- Exporting Layout Design Data — Layout design data, in several industry-standard formats, can be imported to and exported from the Ansys Electronics Desktop (AEDT)™ electronics systems design platform projects for post-processing, simulation, and analysis. Layout design data, in several industry-standard formats, can be imported to and exported from AEDT projects for post-processing, simulation, and analysis.
- Importing Circuit Models — Ansys provides support for imported models and libraries as components in the **Schematic Editor**, including SPICE/PSPICE components and libraries, IBIS buffer elements, Verilog components, W-elements, and X-parameter elements. Frequency-dependent network design data can be imported and used to generate schematic components.
- Exporting Network Design Data — The Linear Network Analysis Circuit solver and the field solver tools produce network design data that can be exported for use by other tools.
- Importing Solution Data — Solution data can be imported into and exported from Ansys projects

Importing Layout Design Data

Design data, in several industry-standard formats, can be imported into the Ansys Electronics Desktop (AEDT)™ electronics systems design platform.

Importing ANF Design Data

Ansys Neutral File (ANF) formatted files are typically generated by third party tools or by the “ALinks” program that translates third-party designs. ANF is a public, neutral file format that allows third party tools to exchange design data with Ansys products. ANFv2 and ANFv4 are two different public, neutral file formats. An ANFv4 file may contain schematic data, 2D geometry (layout), 3D geometry, and component data, but none of the data is required. A given ANFv4 file may contain just schematic data with no layout data, or may contain just layout data with no schematic data.

> **Note:**
>
> If material properties are not provided in the ANF file, the required material properties are taken from the Ansys Electronics Desktop (AEDT)™ electronics systems design platform material library database. These properties may be different from those in the tool used to generate the ANF file.

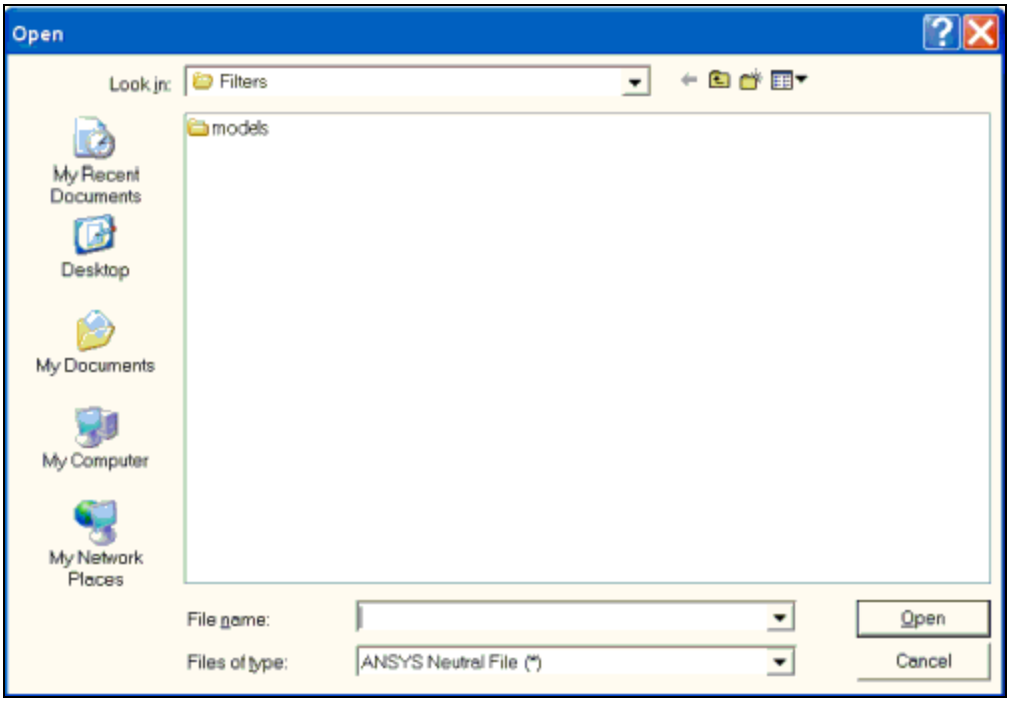
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When geometry is imported from ANF, ports are automatically added, and a subdirectory for the ANF-based project is configured in the Project directory. When ANF-based projects are saved, the **Save As** window always opens, regardless of where the ANF file originated. AEDT can import both ANFv2 and ANFv4 formatted files.

Importing ANFV2 Design Data

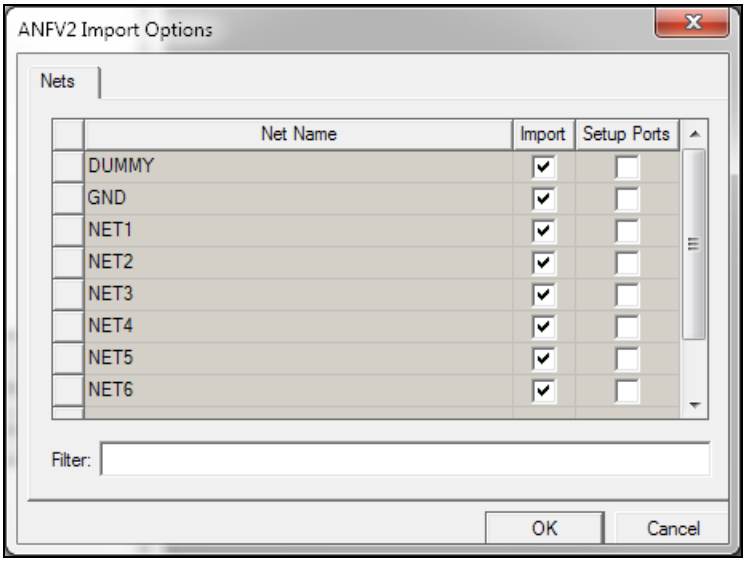
Follow these steps to open and import an ANFVS2-based project.

1. From **File**, select **Import** > **ANF** to open an explorer window.



2. click the **Look in** field to locate the directory containing the file with the project you want to open. Click the **Files of type** field to display the files with the ANF format.
3. Click ANF format file in the window to select it, or type its name into the **File name** field. If you select to open an ANFV2 file, a window appears that allows you to select the nets to import , then generate ports on. You can also select an associated .cmp file to import with it.

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



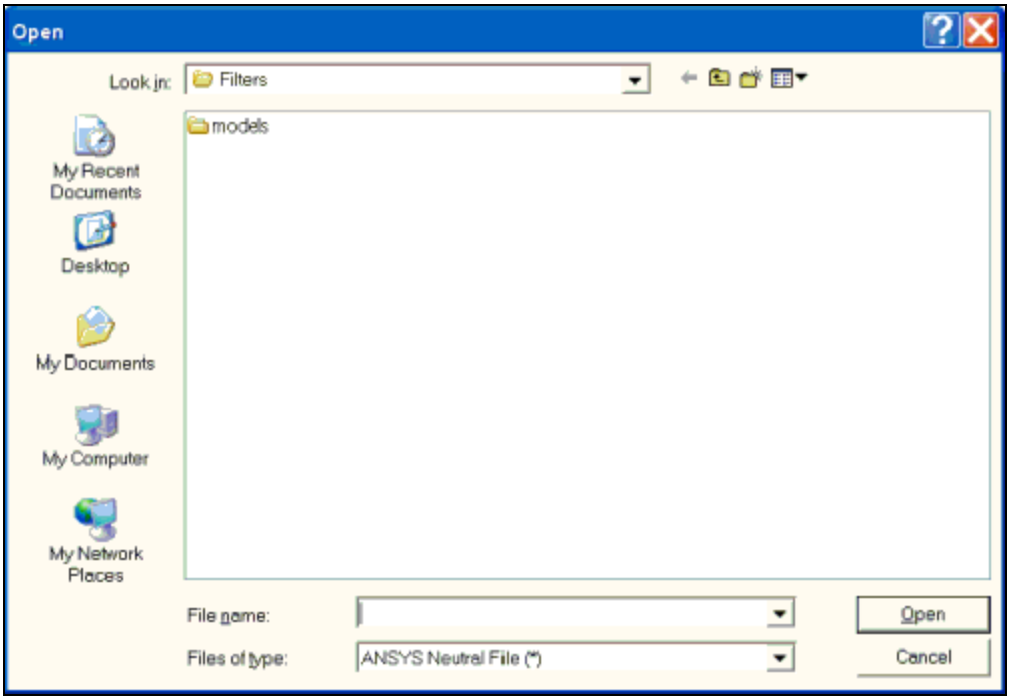
4. When you click **OK** in the **ANFV2 Import Options** window, the file is imported.

Only nets with a check in the **Import** column are imported. Pins that belong to nets with a check in the **SetupPorts** column become ports when imported. The **Filter** control can be used to select which nets are shown in the window.

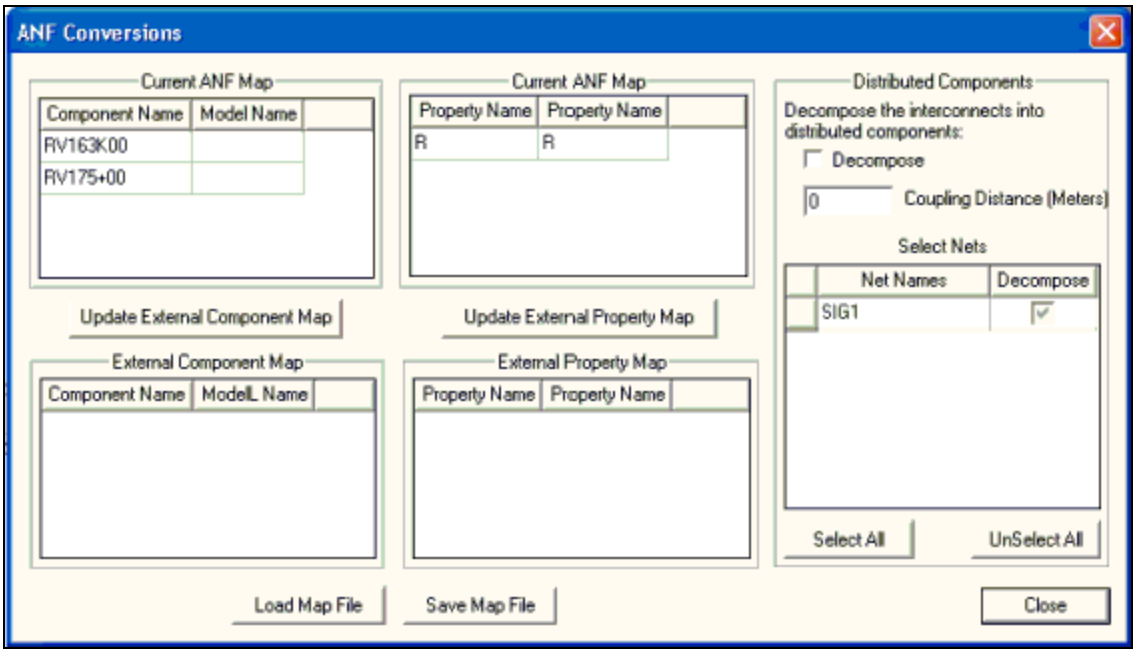
Importing ANFV4 Design Data

1. To open an ANFV4-based project, on the **File** menu, select **Import** > **ANF** to open an explorer 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. click the **Look in** field to locate the directory containing the file with the project you want to open. Click the **Files of type** field to display the files with the ANF format.
3. Click ANF format file in the window to select it, or type its name into the **File name** field.
4. Click **OK**. The **ANF Conversions** 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.

5. The **Current ANF (Components) Map** group boxes list the components on the ANF file and any associated models. You can enter the names of a model manually in the **Property Name** field. The ModelName property for the component are set to the entry in the **Model Name** field in the mapping, and the Netlist property begins with that name instead of the component name.
6. **Update External Component Map** saves any mappings you enter manually to a text file. Click **Save Map File** to open a File explorer window. Use the window to browse to the directory where the map file is to reside, then enter the name of the file. The map file is saved with a **.mapping** extension. The same mapping file should be used for both component and property mappings. After the mapping file has been created, **Update External Component Map** saves any mappings you enter manually.
7. The **Current ANF (Property) Map** group boxes list the mapping of properties in the ANF file to the properties that are used when the file is imported. You can enter the mapped name of a property manually in the **Model Name** field. The name for the property are set to the entry in the netlist.
8. **Update External Property Map** saves any mappings you enter manually to a text file. Click **Save Map File** to open a File explorer window. Use the window to browse to the directory where the map file is to reside, then enter the name of the file. The map file is saved with a **.mapping** extension. After the mapping file has been created, **Update External Component Map** saves any mappings you enter manually.
9. **Load Map File** opens an explorer window. Browse to the directory where the map file resides, then select or enter the name of the file. The map file can contain component and/or property mappings, and must have a **.mapping** extension. The same mapping file should be used for both component and property mappings.
10. The **External Component Map** area lists the components that have been read from an external mapping file. The display changes each time the external file is updated on the window.
11. The **External Property Map** area lists the properties that have been read in from an external mapping file. The display changes each time the external file is updated from the window.
12. The **Distributed Components** area allows you to specify that imported interconnects are converted to distributed components if they fall within a minimum coupling distance (specify distance), and select the nets that are thus converted.
13. Click **Close**. The **Layout Editor** shows the design, and the projects window shows the imported project.

Importing Files into the Layout Editor

Using the **File** > **Import** command, you can read the following files directly into the layout editor.

- EDB, Cadence APC/Allegro/SiP, ANF, AutoCAD, GDSII, IPC2581a, XFL, Gerber, IDF, and ODB++ imports opens directly into a new project.

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

- ANX import opens directly into a selected project. Users select whether to add the import to a new design or to an existing design.

Split plane layers have not been poured (generated). Unused interior pads are not output. Nets have not been defined or have been "anonymized" away. Components lack reference designators or part numbers.

> **Note:**
>
> ODB++ allows the export of a .copper_weight for signal layers but not an actual thickness. Use a conversion factor of 1.4mil per ounce of copper. ODB++ v.8 allows for the export of .dielectric_constant, .loss_tangent and .bulk_resistivity but no material names. Generate the name of the layers by prepending ODB_COND_ and ODB_DIEL_ before the first layer the material properties show up in.

> **Note:**
>
> For the following, designs are imported with only the top and bottom layer visible:ODB++, Cadence APD/Allegro/SiP, and ANFV2.

To import a 2D model file:

1. Start a new project to open the Layout Editor.
2. Click **File** > **Import**, then choose and select the type of file to import from an explorer window.
3. click the **Look in** field to locate the directory containing the file you want to import. Click the **Files of type** field to display the files with the format you want to import.
4. Select the 2D model file you want import on the display.
5. Click **Open** to open a window specific to the requested format.

Modification after Import Stackup Options

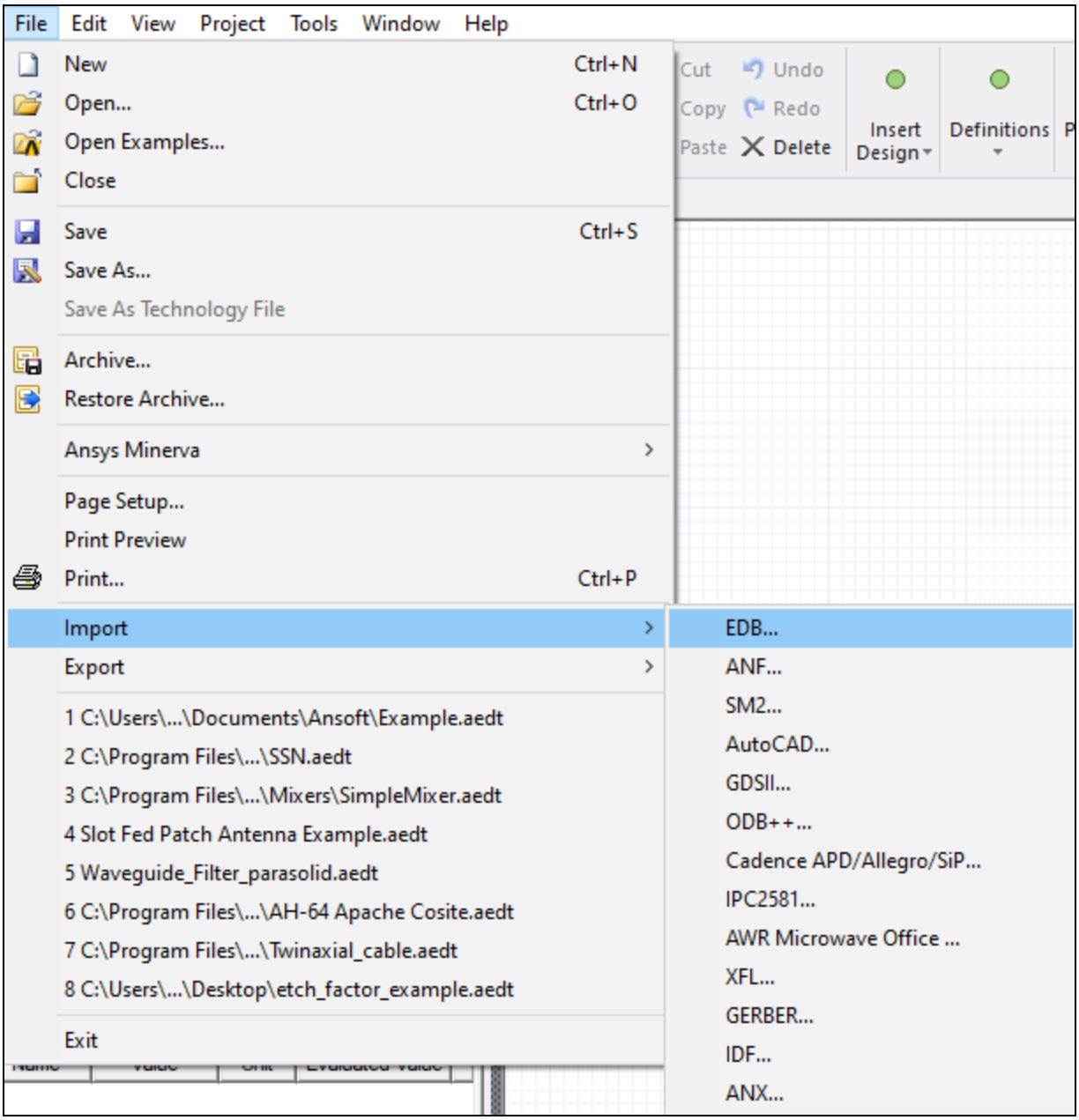
For a detailed description of how to combine exported .xml files, see Layer Stackup window.

Importing EDB Format Files

Follow these steps to import an EDB file, select **File** > **Import** > **EDB** which brings up a file open window.

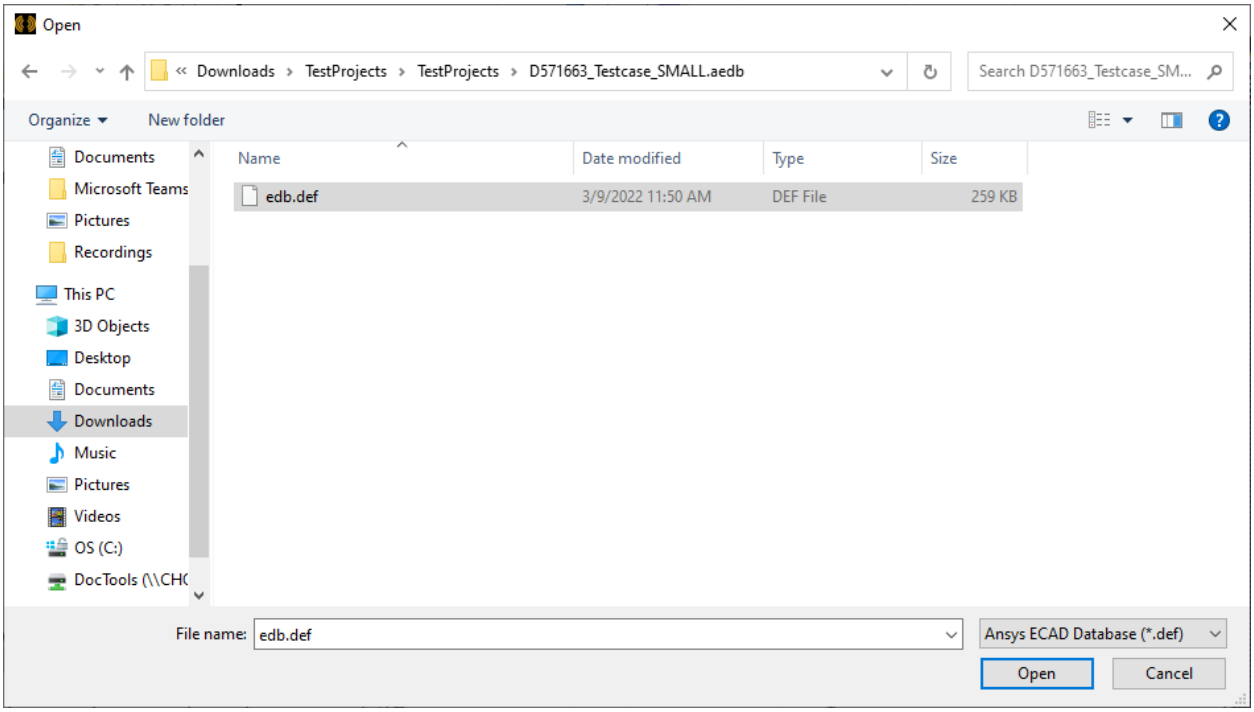
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **File** > **Import** > **EDB** to open an explorer window.



2. Browse to and select the appropriate <database>edb.def (i.e., Ansys ECAD Database) file. Then click **Open** to close the explorer window.

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



A new project and design are created around the layout data found in the AEDB database.

> **Note:**
>
> If the database is already referenced by another the Ansys Electronics Desktop (AEDT)™ electronics systems design platform project, the two projects share the same database. No synchronization is provided for edits made through the independent projects (e.g., if the ports are deleted through one project, they are invalid in the other project and the behavior are indeterminate).

Importing Gerber Format Files into the Layout Editor

Only version RS-274X Gerber designs are supported. Assume the files are contained in either a directory or a directory that has been tar'ed and gzip'ed. On linux use "tar -cvzf gerber.tgz /home/user/gerber", where "gerber" is the name of the directory that contains the design. Support is also included for NC-Drill files (circular holes and drawn slots only).

The window is initialized with the name of the files identified as RS-274X or NC-Drill files. Only the files in the topmost directory are looked at. NC-Drill files are processed separately and their type 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.

- **Import stackup and options** allows you to import information from an xml control file or a layer mapping file into the window.
- **Export control file** creates an xml control file using the information imported to the window to the location specified.

Layers Pane

- **File Layer Name** is name of the file in the directory; it does not include the path.
- **Import check box** specifies which layers to import.
- **Import layer name** is the name of the new layer. Note the information contained in the control file is exported on the **Layer Stackup** window and is not directly usable by the translator — if layer names are changed. One can, however, combine the records for renaming layers with the control file exported on the stackup window; for more information see Control File Usage with Renaming. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform also supports reading .tech and .layermap files.
- **Type** can be changed and identifies the layer type that are used to place each layer in the file. Via and wirebond types are special and allows users to select the start and stop layers by expanding the window's width.

Options Pane (NC Drill)

If your drill holes appear to be out of whack, you need to expressly describe the format of the NC Drill file. Many cad systems include a comment in the NC Drill file that are automatically processed, but if this does not work the parameters can be set explicitly. Typically, both fields need to be set.

- **Format(digit.digit)** – Unit type is always defined but typically decimal points are not used in the file and you need to know how many digits are before the decimal and how many are after (e.g., "2.4"). The period is required and the format is "digit period digit".
- **Omit Zeroes** – Zeroes can be omitted on the leading or trailing side of numbers.

Via and Wirebond Types

- **Via** – By "_Default_", Start Layer is the top-most signal layer and Stop Layer is the bottom-most layer. To set up blind and buried vias it is best to start by setting the layer types for all layers , then reorder by dragging the rows by selecting the filled-in cell on the left.
- **Wirebond** – By "_Default_", Start Layer" and "Stop Layer" are set to the signal layer directly underneath the Wirebond. If no signal layer lies underneath the Wirebond the "Start Layer" and "Stop Layer" are set to the bottom-most layer. Wirebond layer rows can be moved up and down by dragging and dropping.

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

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

ECAD packages (for example, Allegro from Cadence, 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

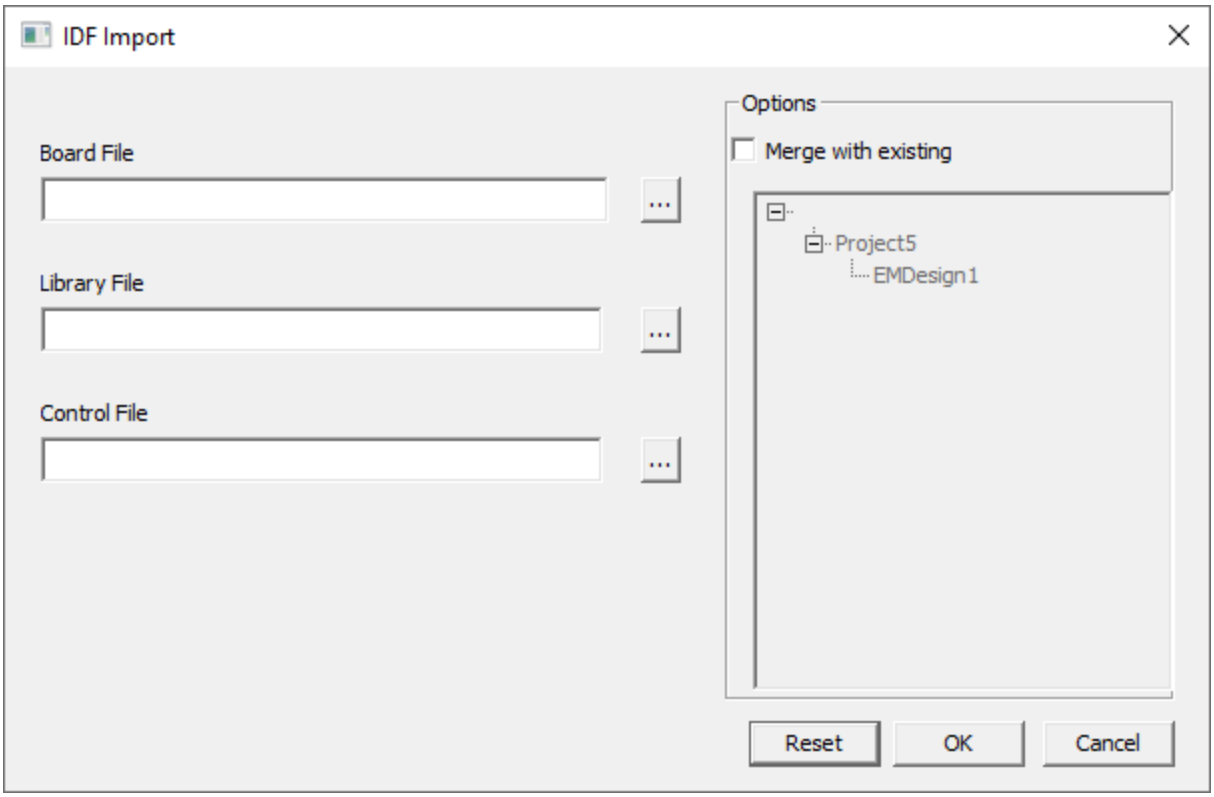
Follow this procedure to import an Intermediate Data Format (IDF) files that have been exported from an ECAD package.

1. Choose **File** > **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, 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 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. If updating components with IDF information:
    a. In the **Options** area, click **Merge with existing**.
    b. Select the existing file.
    This merges any new properties into the existing file. To update a component with IDF data, the component must exist in both the design and IDF files.
6. Click **OK** to import the model.

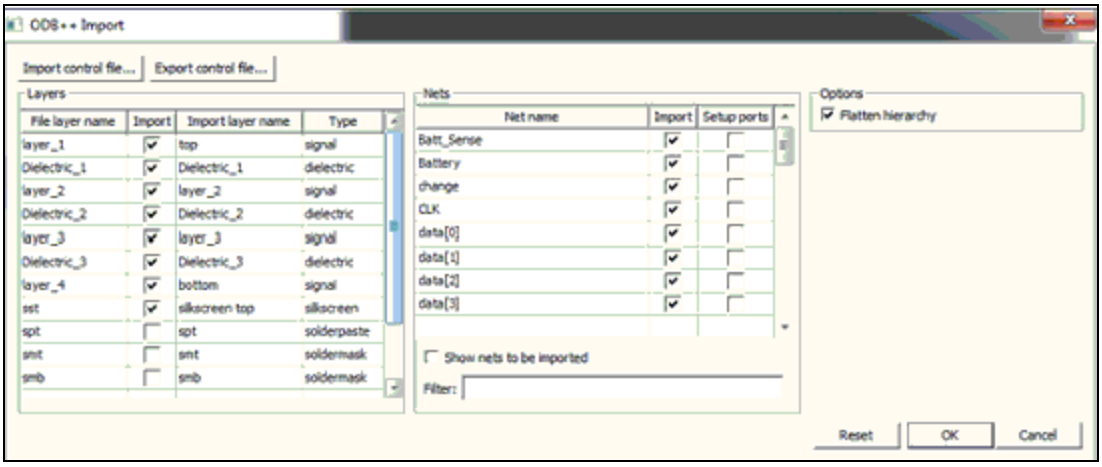
Importing ODB++ into the Layout Editor

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform supports importing ODB++ from many platforms. The translator relies on optional information in the EDA data file and component layer files to integrate the manufacturing data into PCB level objects such as Padstacks. If this data is missing the design may not be imported correctly. When first importing a design into AEDT, please make note of any error or warning messages; common issues include:

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

- Split plane layers have not been poured (generated)
- Unused interior pads are not output
- Nets have not been defined or have been "anonymized" away
- Components lack reference designators or part numbers

To access the window shown in the following diagram, refer to Importing Files into the Layout Editor. Select a *.tgz* file or a Windows directory to import, to open the **ODB++ Import** window.



The window is initialized with the nets and layers on the imported design. Note that while signal layers can be excluded, they are imported if a hole begins or ends on that layer. Excluding layers can speed up both the translation and import into AEDT. Excluding nets can also speed up the import of a design into AEDT and make the design easier to work with.

- **Import stackup and options** allows you to import information from an xml control file or a layer mapping file into the window. Note the information contained in the control file is exported on the Layer Stackup window and is not directly usable by the translator if layer names are changed. One can however combine the records for renaming layers with the control file exported on the stackup window; for more information see Control File Usage with Renaming. Ansys Electronics Desktop also supports reading .tech and .layermap files.
- **Export control file** creates an xml control file using the information imported to the window to the location specified. This file can either be imported and used through the window or used directly with Anstranslator.exe.

Layers Pane

- **Import** check box specifies which layers to import. Note that while signal layers can be excluded they always be imported if the signal layer is at the top or bottom of a hole (padstack).

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

- **Import Layer Name** renames a layer. Note that all layers including drawing layers can be renamed. Once imported into AEDT this mapping are lost and you need to refer to these layers by their new name.
- **Type** is set based on properties in the ODB++ file and can be changed if needed.

Nets Pane

- **Import** check box specifies which nets to import. Ensure **Dummy** is checked if you want to import the metal not assigned to a specific net.
- Check **Setup Ports** to automatically creates ports on imported nets.

Options Pane

- **Flatten Hierarchy** should usually be selected. This makes sure that all the component data is moved down and is ready for simulation.

Control File Usage

Importing a control file into the window preserves not only the layer settings but also the Component property section of the .xml file. This means that if you import the following control file C28 has a value of 4.7nF.

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">
  <Components schemaVersion="1.0">
    <TypeMap>
      <!-- Match C28 exactly -->
      <RefDes RegEx="C28" Type="Capacitor" SC_Value="4.7n"/>
    </TypeMap>
  </Components>
</c:Control>
```

For details on the TypeMap see ODB++ Control File Part Mapping.

Importing IPC2581 rev.a, rev.b, and rev.c Files into the Layout Editor

Note that the Ansys Electronics Desktop (AEDT)™ electronics systems design platform only supports IPC2581 rev.a and rev.b, and interprets the default behavior based on the schema. All references to IPC2581 signify only rev.a or rev.b.

IPC2581 rev.a

Design should be exported as "Full".

IPC2581 rev.b

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

Design should be exported as "User" design with all data selected.

IPC2581 rev.c

Design should be exported as "User" design with all data selected.

Importing Files

For information on how to initially access the IPC2581 import window (**File** > **Import**), see the introductory topic Importing Files into the Layout Editor. Select a *.cvg* or *.xml* file or a Windows directory to open the **IPC2581 Import** window. The window initializes with the nets and layers on the imported design.

- **Import stackup and options** allows you to import information from an xml control file or a layer mapping file into the window. Note the information contained in the control file is exported on the Layer Stackup window and is not directly usable by the translator if layer names are changed. The **Edit Layers** window is not used by the translator and is ignored.
- **Export control file** creates an xml control file using the information imported to the window to the location specified. This file can either be imported and used through the window or used directly with Anstranslator.exe. Note that this can be used to read in an IPC Layer Stackup file and export an .xml file that can be used with an IPC2581 file.

Layers Pane

- **Import** check box specifies which layers to import. Padstacks are shortened or removed entirely as selected. Type is set based on properties in the IPC2581 file and can be changed if needed. The only exception is the "SOLDERMASK" layer which are imported as a dielectric layer.
- **Import Layer Name** renames a layer. Note that all layers including drawing layers can be renamed. Once imported into AEDT, this mapping are lost and you need to refer to these layers by their new name.

Nets Pane

- **Import** check box specifies which nets to import. Ensure **Dummy** is checked if you want to import the metal not assigned to a specific net.
- Check **Setup Ports** to automatically creates ports on imported nets.

Importing DXF and DWG Format Files into the Layout Editor

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform imports AutoCAD version 2.5 (AC1002) through 2012 (AC1024) on Linux and version 2.5 (AC1002) through 2017 (AC1027) on Windows. See the introductory topic Importing Files into the Layout Editor for the initial steps in the process of importing 2D data into the Layout Editor. Once a 2D model file is selected, an explorer window opens.

The window is initialized with the layers on the imported AutoCAD file.

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

- **Import stackup and options** allows you to import information from an xml control file or a layer mapping file into the window.
- **Export control file** creates an xml control file using the information imported to the window to the location specified.

Layers Pane

- **File Layer Name** is name of layer as stored in the dxf or dwg file.
- **Import** check box specifies which layers to import.
- **Import layer name** is the name of the new layer. Note the information contained in the control file is exported on the **Layer Stackup** window and is not directly usable by the translator — if layer names are changed. One can, however, combine the records for renaming layers with the control file exported on the stackup window; for more information see Control File Usage with Renaming. AEDT also supports reading .tech and .layermap files.
- **Type** can be changed and identifies the layer type that are used to place each layer in the file. Via and wirebond types are special and allows users to select the start and stop layers by expanding the window's width.

Options Pane

- Use **Override AutoCAD file units and the Units** drop-down menu to select units to interpret the DXF/DWG file contents.
- **Import Only Named Nets** – Only applicable when a valid 3D Layer stackup is imported. When checked this option sets *ImportDummyNet="false"*.

Objects Pane

Use the check boxes to fine-tune the import:

- **Auto detect closure** checks polylines to see whether they are closed. If a polyline is closed, AEDT creates a polygon in the design.
- **Self stitch** causes multiple straight line segments to be joined to form polylines. Tolerance is used to decide if two coordinates are the "same" and should be considered for joining. If the resulting polyline is closed, a polygon is created in AEDT. You have the choice of two different tolerances for self stitching: **Default Tolerance** and explicit **Tolerance**. Default tolerance is a geometric tolerance based on the extents of the project being imported. Explicit tolerance allows you to expand the tolerance based on your knowledge of the dxf project being imported.
- **Defeature** removes certain small features in the imported geometry to reduce complexity. The features that are removed include multiple points placed in the specified distance, thin or narrow regions ("thins" and "spikes"), and extraneous points along straight line segments. Specify the distance in the **Tolerance** area.
- **Round coordinates** rounds all imported data to the specified number of decimal places.

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

- **Convert closed wide lines to polygons** imports wide polylines as polygons. You have more flexibility to change the shape of such an object when it is imported as a polygon.

The following DXF entities can be imported as two-dimensional objects from AutoCAD:

- Arc
- Solid
- Circle
- Block
- Ellipse
- Array of blocks (MInsert)
- 2D Polyline, Polyline, and Line

When you have completed selections on both tabs, click **OK** on either tab and the file is imported into the active **Layout** window.

Via and Wirebond Types

- **Via** – By "_Default_", Start Layer is the top-most signal layer and Stop Layer is the bottom-most layer. To set up blind and buried vias it is best to start by setting the layer types for all layers , then reorder by dragging the rows by selecting the filled-in cell on the left.
- **Wirebond** – By "_Default_", Start Layer" and "Stop Layer" are set to the signal layer directly underneath the Wirebond. If no signal layer lies underneath the Wirebond the "Start Layer" and "Stop Layer" are set to the bottom-most layer. Wirebond layer rows can be moved up and down by dragging and dropping.

Importing Cadence APD/ALLEGRO/SIP Files into the Layout Editor

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform supports Extracta Import from Cadence. Note that Extracta.Exe is a Cadence supplied executable and must be installed on your machine and on your executable path for this to work. To access the window shown in the following diagram, refer to Importing Files into the Layout Editor. Once a *.brd*, *.sip*, or *.mcm* file is selected, an explorer window opens.

The window is initialized with the nets on the imported design.

- **Import stackup and options** allows you to import information from an xml control file or a layer mapping file into the window. Note the information contained in the control file is exported on the Layer Stackup window and is not directly usable by the translator if layer names are changed. One can however combine the records for renaming layers with the control file exported on the stackup window; for more information see Control File Usage with Renaming. Ansys Electronics Desktop also supports reading .tech and .layermap files.

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

- **Export control file** creates an xml control file using the information imported to the window to the location specified. This file can either be Imported and used through the window or used directly with Anstranslator.exe.

Nets Pane

- **Import** check box specifies which nets to import.
- Check **Setup Ports** to automatically creates ports on imported nets.

Options Pane

- **Import Dummy Net** – Imports all metals on the specified net. This is independent of the list of nets.
- **Use Default Component Values** – If components come in without resistor, capacitor or inductor types, a default RLC value is set commensurate with the type and a best guess at pin pairing is generated.
- **Import Hatching** – Extracta has an option to export plane data, with or without hatching. If this option is checked hatching are generated.
- **Import Degassing Holes** – Degassing holes defined in the source plane or mask data are imported into the layout.
- **Mask Layer Import Options** – Opens the **Mask Layer Geometry Import Options** window. Either select **Import all mask layer geometry** or **only shield layer geometry**. If **only shield layer geometry** is selected, either select **Use the default subclass names** or select **Use custom names** and enter a unique name for the **Top** and **Bottom shield layers**. Refer to Importing a Multizone Stackup.
- **Import Bondwire Definitions** – Select an XML file from which to import bondwire definitions

Control File Usage

Importing a control file into the window preserves not only the layer settings but also the Component property section of the .xml file. This means that if you import the following control file, C28 has a value of 4.7nF.

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">
  <Components schemaVersion="1.0">
    <TypeMap>
      <!-- Match C28 exactly -->
      <RefDes RegEx="C28" Type="Capacitor" SC_Value="4.7n"/>
    </TypeMap>
  </Components>
</c:Control>
```


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

For details on the TypeMap see ODB++ Control File Part Mapping.

Importing XFL Files into the Layout Editor

For information on how to initially access the XFL import window (**File** > **Import**), see the introductory topic Importing Files into the Layout Editor. After selecting an *.xfl* file, the XFL import window opens initialized with nets on the design.

- **Import stackup and options** allows you to import information from an xml control file or a layer mapping file into the window.
- **Export control file** creates an xml control file using the information imported to the window to the location specified. This file can either be imported and used through the window or used directly with Anstranslator.exe. Note that this can be used to read in an IPC Layer Stackup file and export a *.xml* file that can be used with an IPC2581a file.

Nets Pane

- **Import** check box specifies which nets to import.
- Check **Setup Ports** to automatically creates ports on imported nets.

Importing GDSII Format Files in IC Mode

GDSII files are imported into IC mode through the GDSII import window. This window lists layers in the design, allowing users to designate a control file (e.g., a technology file), select from several options, and/or import information from an XML control file, layer mapping file, or a layer filter. Once GDSII files (*.*gds*, *.*sf*, or *.*strm*) are imported into the Ansys Electronics Desktop (AEDT)™ electronics systems design platform, they can be edited in the **Layout Editor** (IC mode). IC mode reconfigures the **Layout** tab with new commands which allow users to import, simplify, and convert large GDSII files into EDB (i.e., Electronic Database) files.

1. Import the GDSII file with a control file.
2. Simplify the design using the tools available in IC mode:
    a. Crop and delete inappropriate objects
    b. Group vias
    c. Snap vias
    d. Wrap geometry
    e. Snap primitives
    f. Trace nets

Importing a GDSII file

The **GDSII Import** window provides a basic layer mapping which is numerically ordered based on the first layers provided in the GDSII file. To improve the mapping, add a control or technology file, and a layer map file. These files map layers in the GDSII file to layers in the

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

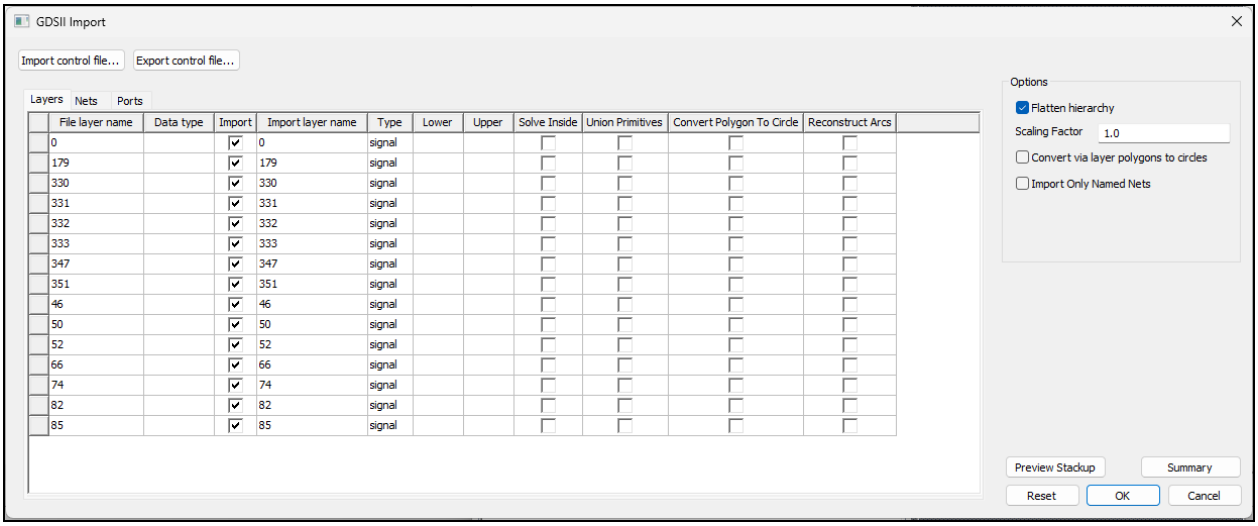
imported stackup. Each set of geometries in the GDSII file only appear on a single layer, so multiple file layers can map to an import layer but file layers cannot map to multiple import layers.

Layers can be renamed during the import process, but the information contained in the control file is exported on the Layer Stackup window and is not directly usable by the translator. When a design is imported, layer mapping is not maintained internally. To re-translate a design with renamed layers, use the same layer names specified in the GDSII file. It is also possible to combine the records for renaming layers with the control file exported on the stackup window. See Control File Usage with Renaming.

> **Note:**
>
> A new OASIS translator (i.e., Open Artwork System Interchange Standard) has been introduced as a beta feature in Anstranslator. To use this functionality, enter `-i=oasis` at the command prompt while running a command. All arguments that are valid for GDSII files should work with OASIS (*.oasis) files. The **GDSII Import** window in AEDT now supports importing OASIS files.

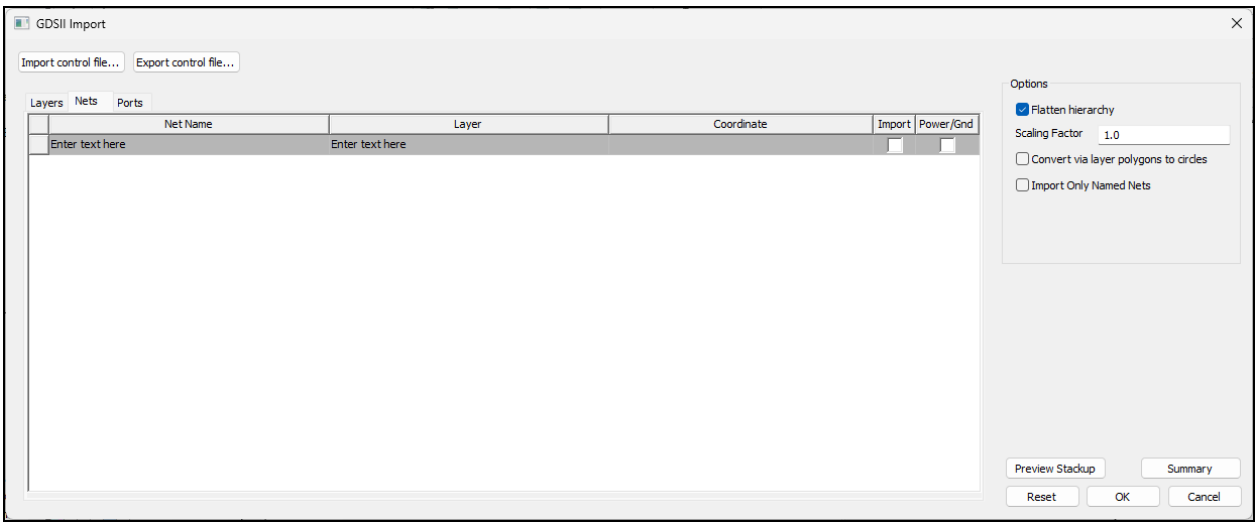
If all conductive layers have thickness, the three-dimensional connectivity of the design is used to generate nets. Follow these steps to import a GDSII file with AEDT.

1. From the **File** menu, select **File** > **Import** > **GDSII** to open an explorer window.
2. Navigate to the selected GDSII (*.gds) file in the explorer window. Then double-click the file or select the file and click **Open** to open the **GDSII Import** window, on the **Layers** tab. This window contains three tabs: **Layers**, **Nets**, and **Ports**.
    - **Layers** – displays layer import options, layer parameters, and editable geometry options (i.e., **Solve Inside**, **Union Primitives**, **Convert Polygon To Circle**, and **Reconstruct Arcs**).

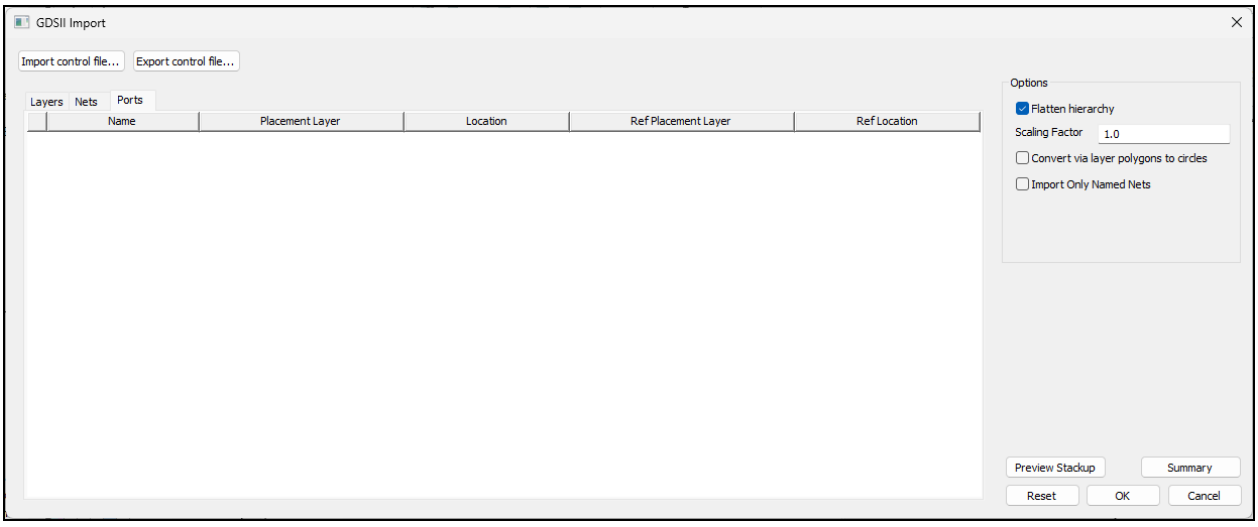


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Nets** – allows users to select nets by **Net Name**, **Layer**, or **Coordinate** for **Import** and designate **Power/Gnd** nets.



- **Ports** – provides a list of ports and a method of adding ports via *.csv* file. Refer to Add Ports From a *.CSV* File.

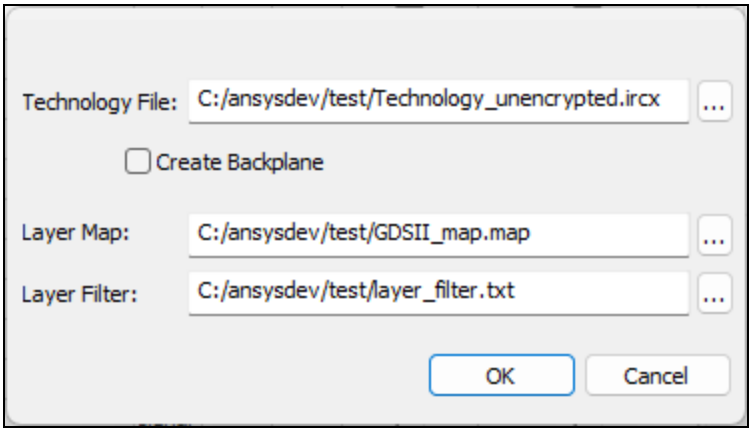


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

During GDS/XML import, auto-component creation prevents overlapping or duplicate pin geometries. Primitives on auto component layers are clustered by physical connectivity, and pins are created only on the smallest area primitive in each connected group. Duplicate geometries are removed and warning messages are issued when overlaps are detected. Component creation always succeeds but redundant pins are not added.

3. Use an XML control file to change import settings. To import a control file or technology file, perform the following substeps:
    a. From the **GDSII Import** window, click **Import control file** to open a window with three fields (i.e., **Technology File**, **Layer Map**, and **Layer Filter**).



Note:

A **Technology File** and **Layer Map** are required. All selected files **must** reference compatible (i.e., in sync) data or errors will result.

    b. Click the **[...]** button next to the **Technology File** field to open an explorer window.

Note:

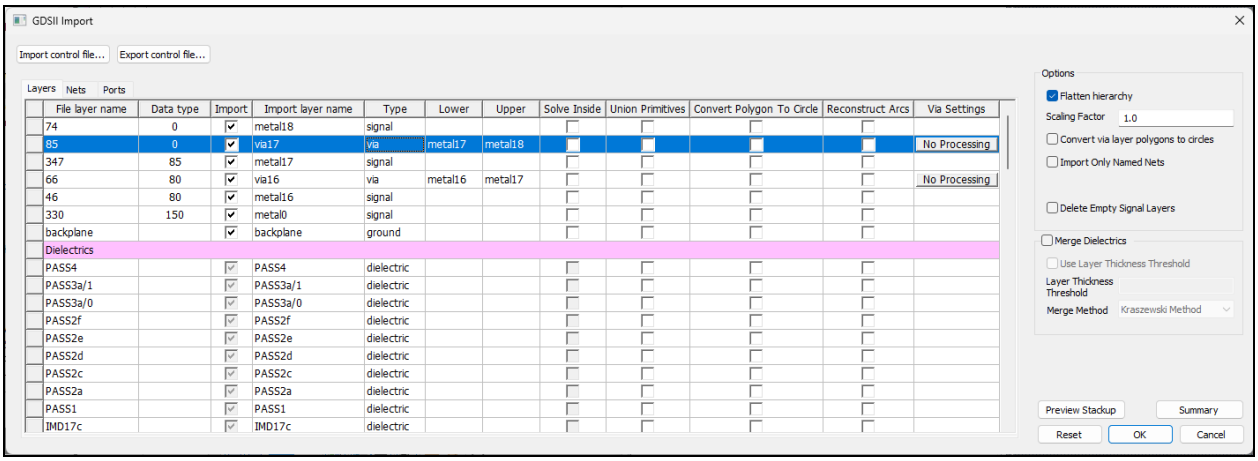
To change the root directory that opens when clicking the **Technology File** **[...]** button, create the environment variable `ANSYS_GDS_TECHFILE_ROOT` with the value of the appropriate directory path (e.g., `C:\TechFiles\TestFolder`).

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

c. Navigate to the selected technology file or control file. Then double-click the file or select the file and click **Open** to close the explorer window and return to the previous window.

> **Note:**
>
> A technology file can have an *.xml*, *.layermap*, *.ircx*, *.vlc.tech*, or *.itf* extension.

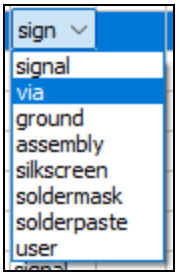
4. If a technology file is imported, at user discretion, check the **Create Backplane** box.
5. If a technology file is imported, click the **[...]** button next to the **Layer Map** field to open an explorer window.
6. Navigate to the selected *.map* file. Then double-click the file or select the file and click **Open** to close the explorer window and return to the previous window.
7. If applicable, click the **[...]** button next to the **Layer Filter** field to open an explorer window. A technology file must either contain references to the same number of signal/via layers in a design's stackup or a layer filter can be used to specify which layers from the technology file to redefine within the stackup. Refer to Adding a Technology File to a Design.
8. Navigate to the selected *.txt* file. Then double-click the file or select the file and click **Open** to close the explorer window and return to the previous window.
9. Click **OK** to close the window and import the technology file and/or the map file. When an imported control file contains dielectric layers, the layers are separated in the **Layers** tab, and an additional **Merge Dielectrics** area appears beneath the **Options** area.



10. If applicable, check the **Merge Dielectrics** area box, and the **Use Layer Thickness Threshold**check box. If the **Use Layer Thickness Threshold** box is checked, enter a value in the **Layer Thickness Threshold** field and select a **Merge Method** from the drop-down menu. Refer to Dielectric Simplification (Merging).

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 applicable, select **Delete Empty Signal Layers**.
12. From the **Layers** table:
    a. Check the boxes in the **Import** column to choose which layers to import (all layers checked by default).
    b. Rename any layer on the text fields in the **Import layer name** column.
    c. Set the type of each layer from the drop-down menus in the **Type** column.



> **Note:**
>
> If a layer's type is set to **via**, the **Via Settings** column appears in the **Layers** tab and the **GDSII Import** window automatically resizes to fit. The geometry on the via layer is converted to a padstack instance between the signal layer above (**Upper**) and the signal layer below (**Lower**). Values for **Upper** and **Lower** are filled automatically.
>
> For on-chip structures, click **No Processing** in the **Via Settings** column to open the **Via Settings** window. Then, at user discretion, check the **Via Grouping** or **Via Snapping** boxes, select **Grouping/Snapping Methods** from the drop-down menus, enter **Tolerance** thresholds, and make additional changes, as appropriate (i.e., check the **Check Containment** or **Remove Dangling Vias** boxes). Finally, click **Apply**.

    d. If applicable, use the check boxes for each layer to enable any/all of the following geometry options:
        - **Solve Inside** – completes simulation calculations within the specified layer.
        - **Union Primitives** – unites overlapping primitives on the same layer.
        - **Convert Polygons to Circles** – converts polygons within a minimum threshold into circles.
        - **Reconstruct Arcs** – smooths and reconnects segmented arcs within a minimum threshold.

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

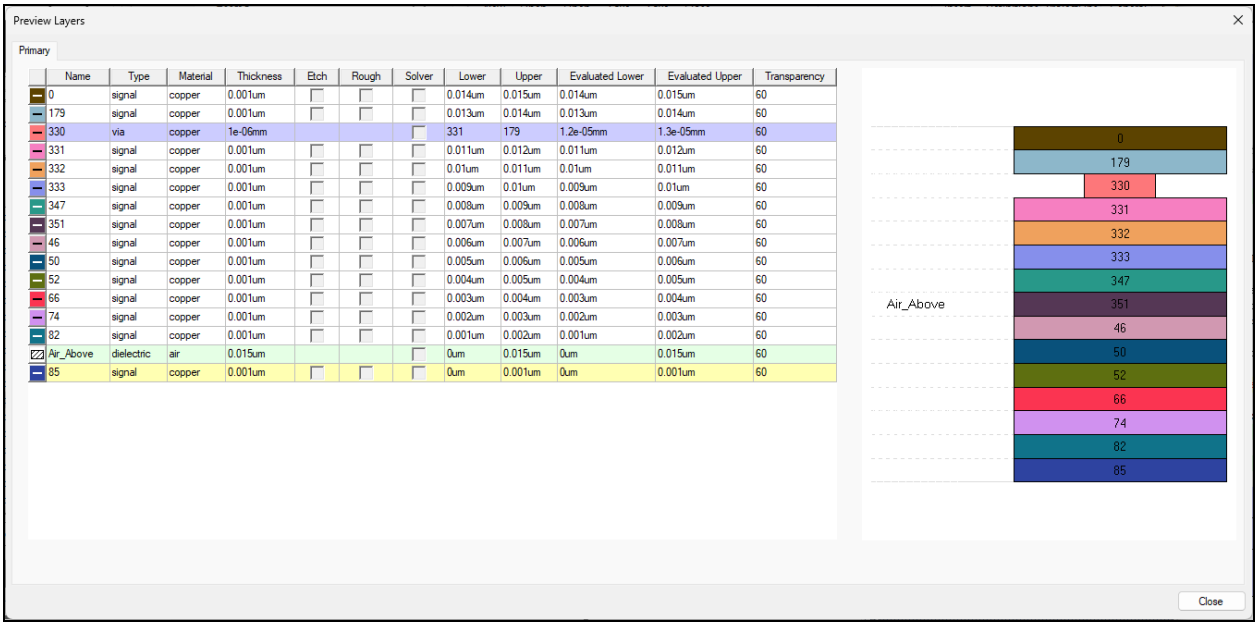
13. If applicable, enable other options in the area:
    - **Flatten hierarchy** – flattens hierarchies in a design *after* the design is modified.
    - **Scaling Factor** – scales the XY coordinates of all geometries. It does not apply to layer thickness (default is 1.0, which represents no change). The scaling factor *t* can be any real number greater than 0.
    - **Convert via layer polygons to circles** – changes any polygons on via layers to circles.
    - **Import Only Named Nets** – only imports geometry that is assigned to a named net.
14. To create a new control file that includes any changes made in this window, click **Export control file.**.
15. Click **OK** to open the file in the **Layout Editor**.

> **Note:**
>
> Any geometrically identical primitives that meet the requirements will be automatically compressed to improve system and simulation performance.

The procedure is complete.

The Preview Stackup Window

Click **Preview Stackup** to open the **Preview Layers** window, which displays the stackup data table and visual representation of the stackup.



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

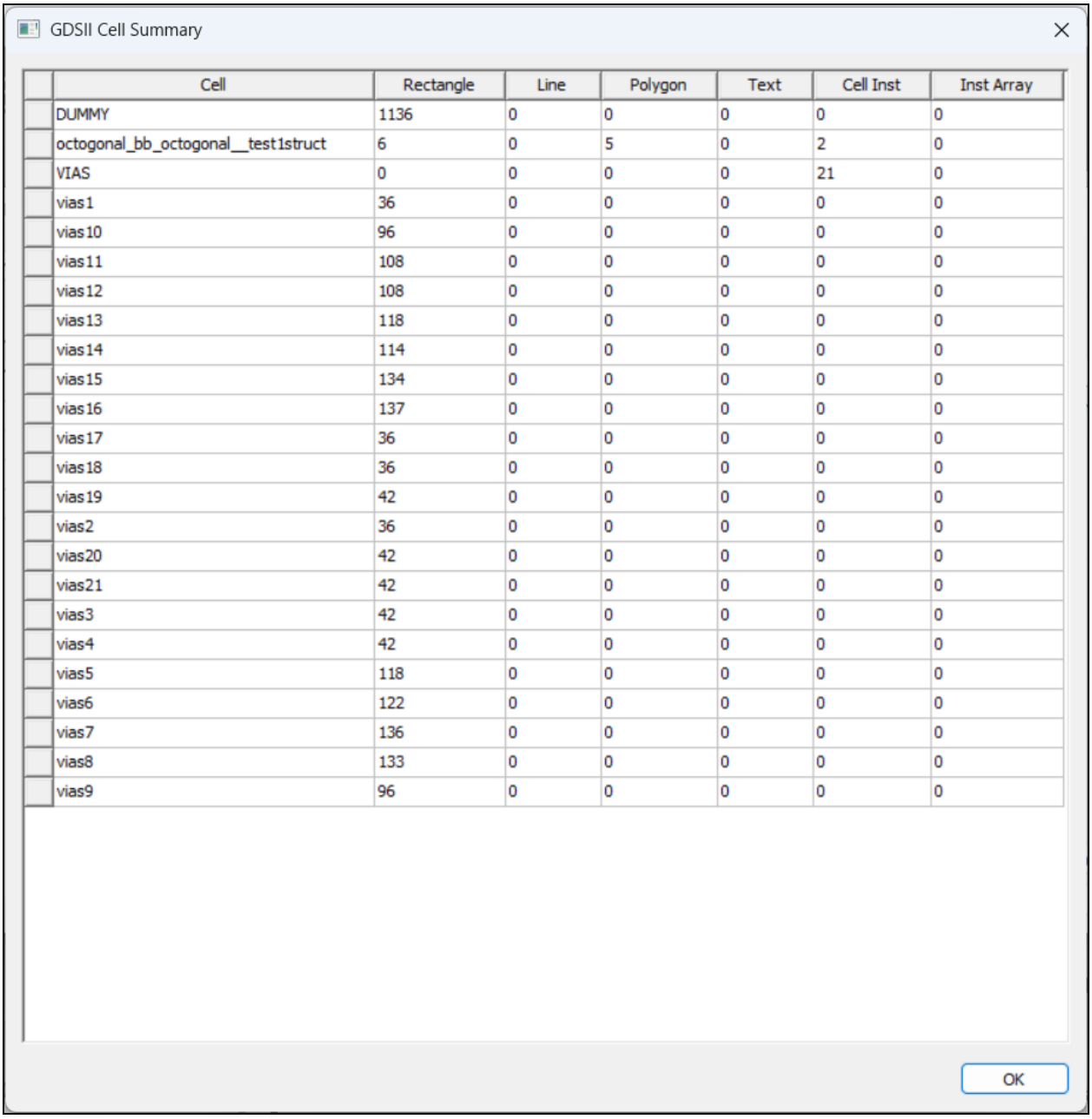
Note:

The default **Thickness** of imported layers is **1e-6m** (i.e., **0.001um**). **Elevation** is calculated dynamically according to the entered thickness of all layers.

The Cell Summary Window

Click **Summary** to open the **GDSII Cell Summary** window, which lists the number of rectangles, lines, polygons, texts, cell instances, and instance arrays for each cell.

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



Cell	Rectangle	Line	Polygon	Text	Cell Inst	Inst Array
DUMMY	1136	0	0	0	0	0
octogonal_bb_octogonal__test1struct	6	0	5	0	2	0
VIAS	0	0	0	0	21	0
vias1	36	0	0	0	0	0
vias10	96	0	0	0	0	0
vias11	108	0	0	0	0	0
vias12	108	0	0	0	0	0
vias13	118	0	0	0	0	0
vias14	114	0	0	0	0	0
vias15	134	0	0	0	0	0
vias16	137	0	0	0	0	0
vias17	36	0	0	0	0	0
vias18	36	0	0	0	0	0
vias19	42	0	0	0	0	0
vias2	36	0	0	0	0	0
vias20	42	0	0	0	0	0
vias21	42	0	0	0	0	0
vias3	42	0	0	0	0	0
vias4	42	0	0	0	0	0
vias5	118	0	0	0	0	0
vias6	122	0	0	0	0	0
vias7	136	0	0	0	0	0
vias8	133	0	0	0	0	0
vias9	96	0	0	0	0	0

Reset

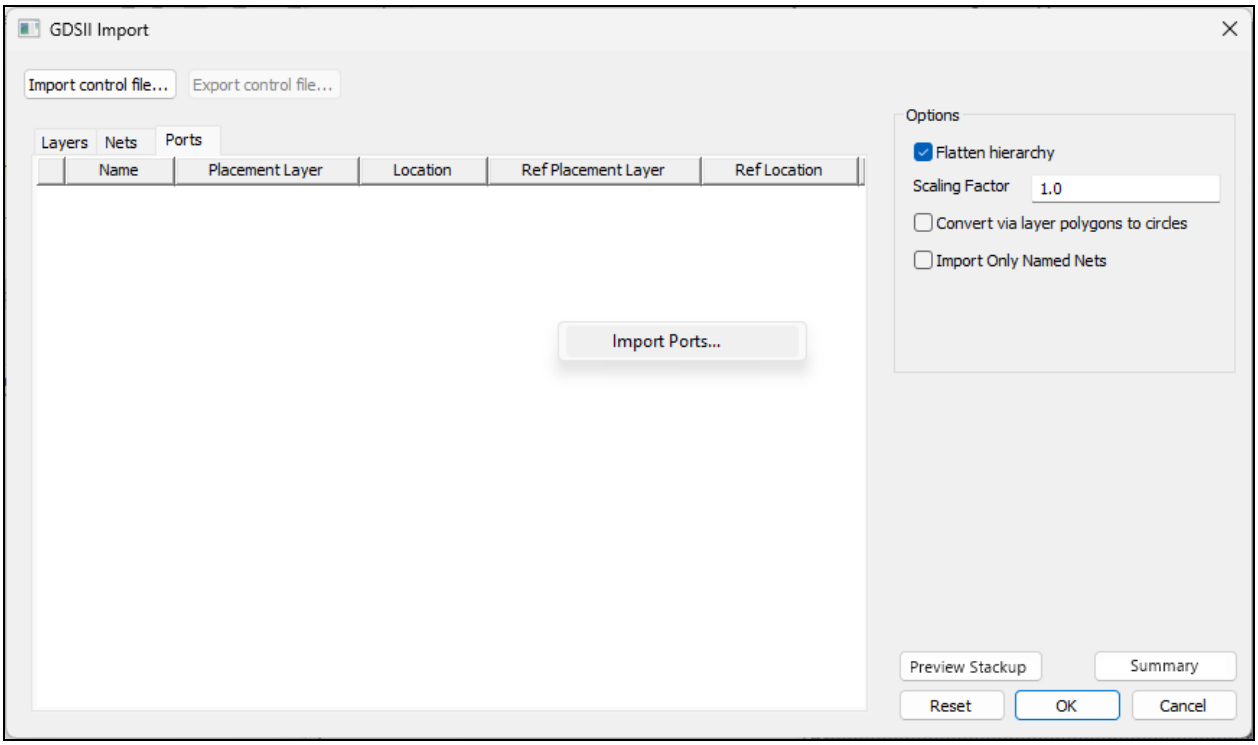
Click **Reset** to undo all changes made since importing the GDSII file.

Adding Ports From a *.CSV* File

Import ports from a *.csv* file by following these steps.

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 **Ports** tab, right-click in the port grid and select **Import Ports** to open an explorer 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:

Here is an example of port data displayed in a file:

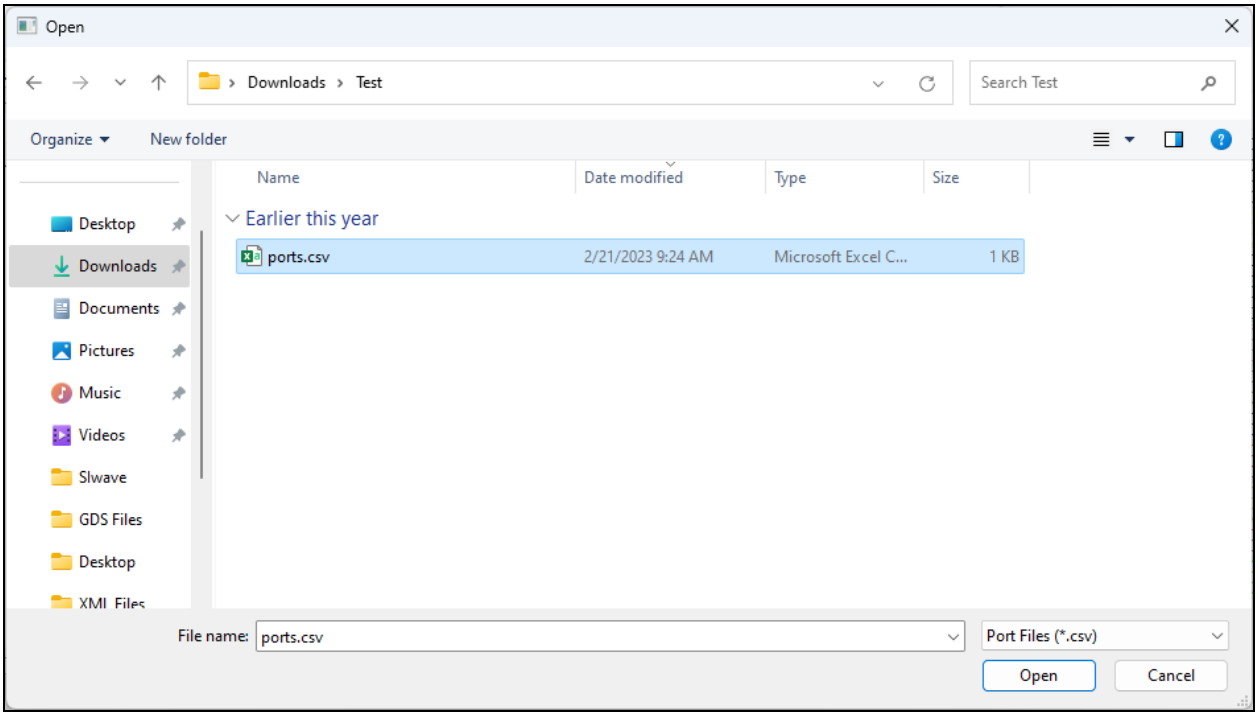
```
! example of two ports
#unit mm
port1  -9 0   Metal9   -9 0   Metal6   40
port2   9 0   Metal9    9 0   Metal6   60
```

Keep in mind the following:

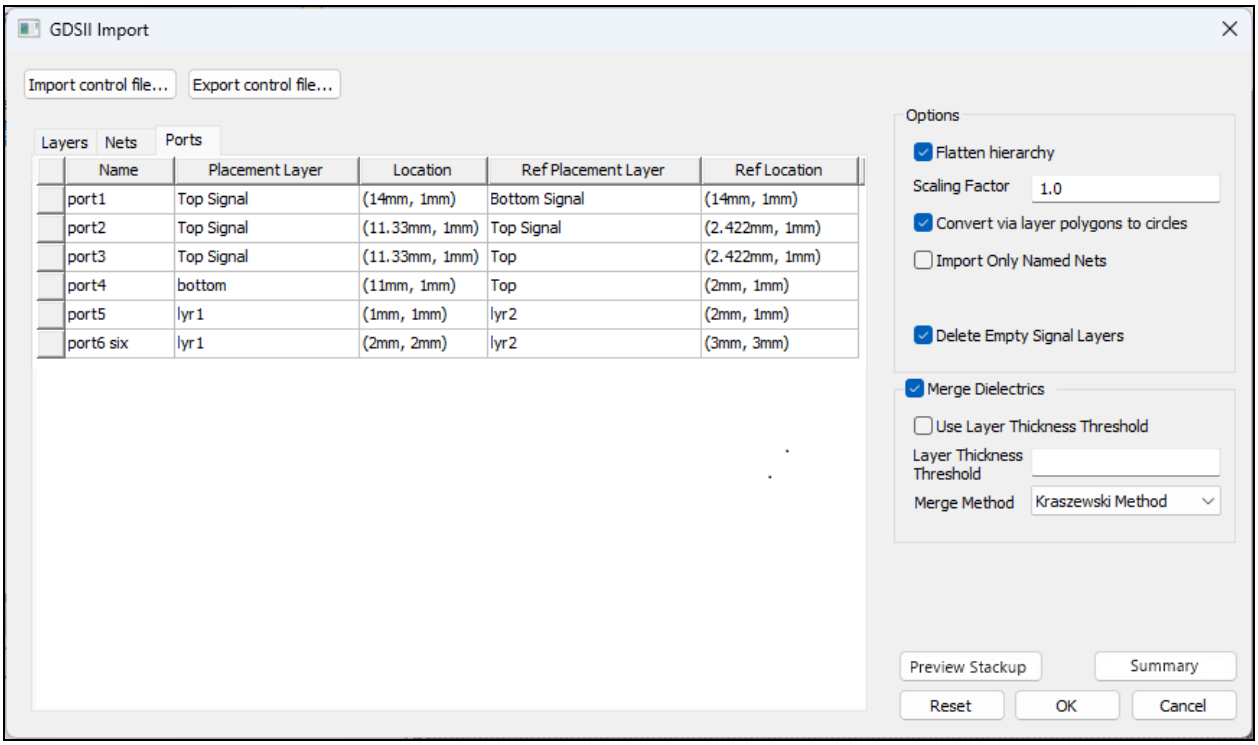
- Port data in a **.csv* file must be entered precisely: <`port name`> <`x coordinate`> <`y coordinate`> <`placement layer name`> <`x coordinate`> <`y coordinate`> <`reference placement layer name`> <Z_0 (i.e., port impedance)>
- `!` and/or `*` are comment lines, ignored during import
- `#unit` <`abbreviation of unit of measurement`> specifies the unit of measurement used
- data can be separated by commas or spaces

2. Navigate to the selected *.csv* file. Then double-click the file or select the file and click **Open** to close the explorer window and return to the **GDSII Import** 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. The ports grid is now populated with the data on the *.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.

4. Click **OK** to save changes and import the GSII file.

The procedure is complete.

Net Tracing in IC Mode

The GDSII file format does not directly support the concept of nets. When the Ansys Electronics Desktop (AEDT)™ electronics systems design platform imports a GDSII file, it creates nets based on the placement of geometries, cells, cell references, and input from the GDSII file's control file text (e.g., text in `GDS_CELL_RULES > BLACK_BOX_CELLS and GDS_NET_DEFINITIONS`). The **Net Tracing** feature uses the control file text to populate the **Net Tracing** window.

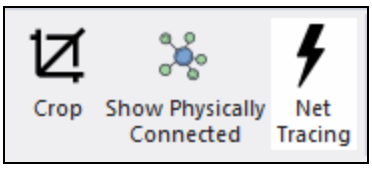
Net Tracing identifies the primitive that encapsulates the text's location (i.e., the seed text), then searches for any primitives that are physically connected to the seed text and adds them to the net. Then Net Tracing searches for primitives that are physically connected to those newly added primitives. This process continues until all physically connected primitives are identified and added to the net.

In addition to text-based tracing, the control file also supports automatic tracing of the **N largest nets** in a design. Largest nets are determined by the total area of physically connected primitives. This option is configured in the `<GDS_NET_DEFINITIONS>` tag using the **AUTO_NETS** attribute, with an optional **AUTO_NETS_PREFIX** to define the naming convention. Auto nets are traced in descending order of size and can be combined with text-based rules (e.g., trace all nets matching `VSS_*`, then trace the 10 largest nets).

> **Note:**
>
> Only the control file can provide seed text for net tracing. New seed text cannot be added in IC mode.

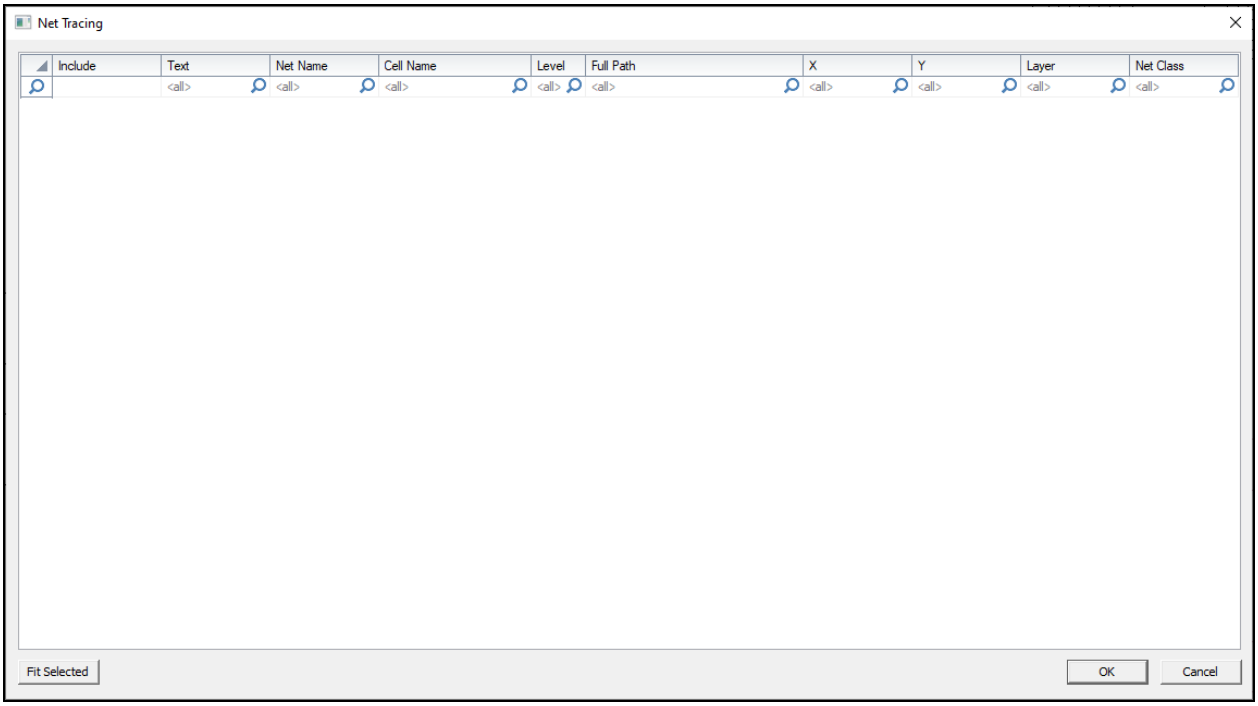
Follow these steps to access the **Net Tracing** window.

1. From the **Layout** tab, click **Net Tracing**.



   The **Net Tracing** window opens and displays a list of all potential nets and columns of relevant data.

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



- **Include** - whether net tracing is performed on this text object (i.e., check or uncheck)
- **Text** - origin text for the net
- **Net Name** - name applied to the net if/when it is created
- **Cell Name** - name of the cell the text is in
- **Level** - level of the net in the design hierarchy
- **Full Path** - specific location of the net in the design hierarchy
- **X** - x coordinate of the starting text
- **Y** - y coordinate of the starting text
- **Layer** - layer the starting text is on
- **Net Class** - name of a group of associated nets

2. Decide which nets to create and check the adjacent box in the **Include** column for each.
3. Click **OK** to close the window. All Included nets are created and their starting text is deleted from the cell. Any remaining text can be used again.

Viewing Physically Connected Elements in IC Mode

From the **Layout Editor**, users can view elements that are physically connected in a design.

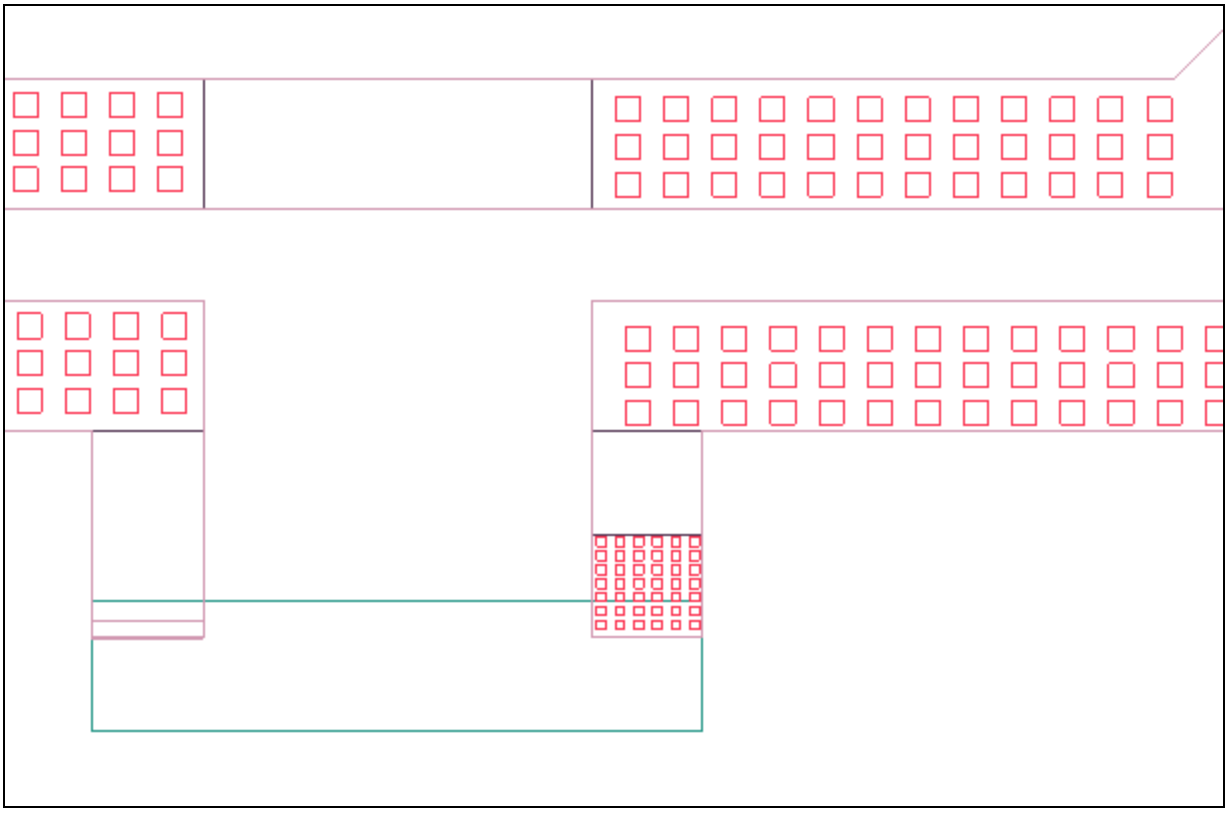
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Not all elements work with the **Show Physically Connected** feature. Connections through solderballs, planar bondwires, and very simple instances of 3D-extrusion are supported. Bondwires with 3D geometry and most 3D-extrusions cannot be tracked.

Complete the steps in the following subsections to view physically connected elements in the **Layout Editor**, and then an organized list of physically connected elements in the **Design List** window.

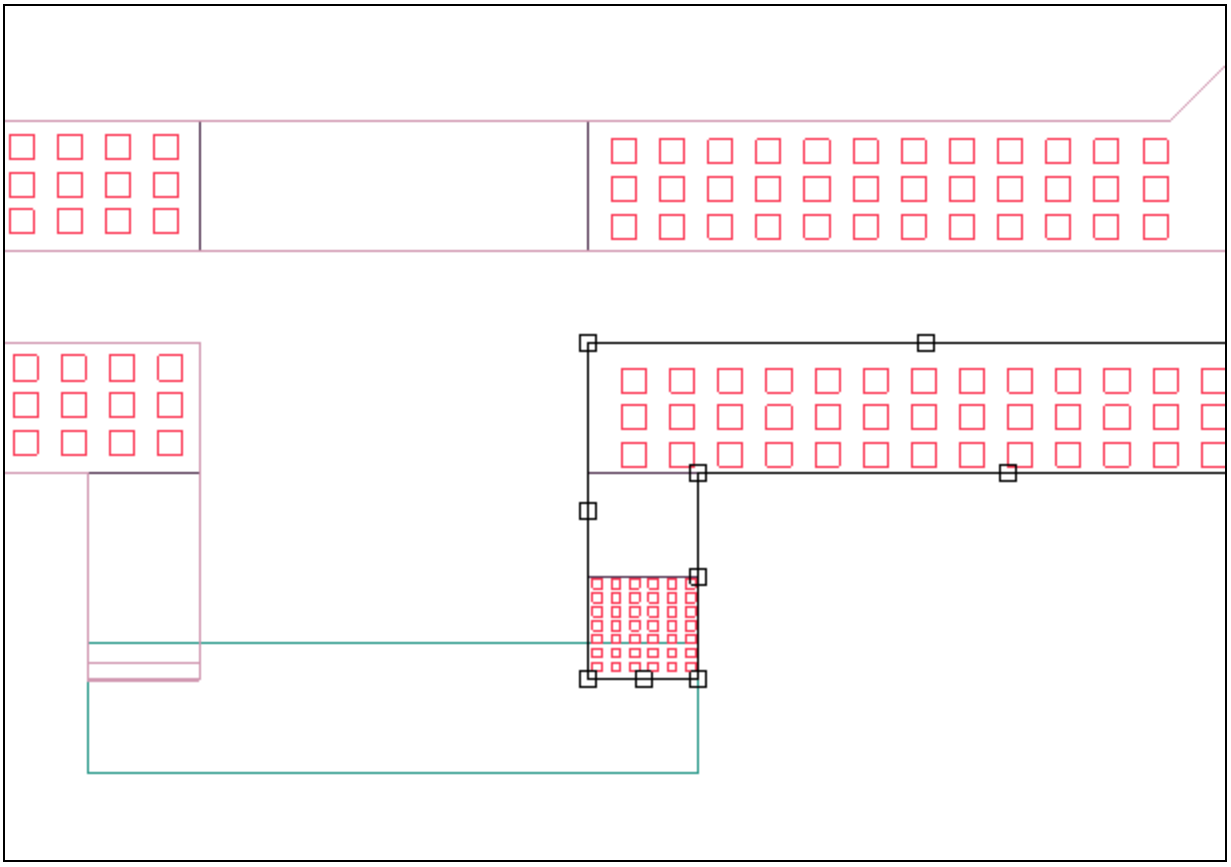
Viewing Physically Connected Elements in the Layout Editor

1. If applicable, navigate to the **Layout Editor** tab and the selected components.

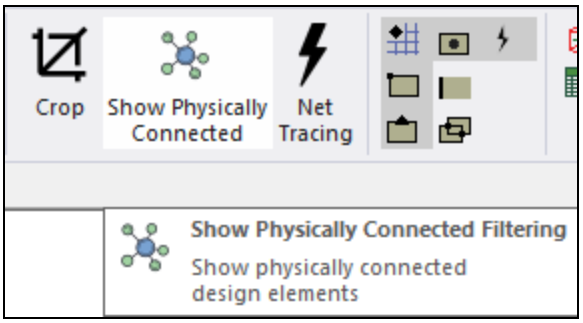


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the **Layout Editor**, select a selected element.

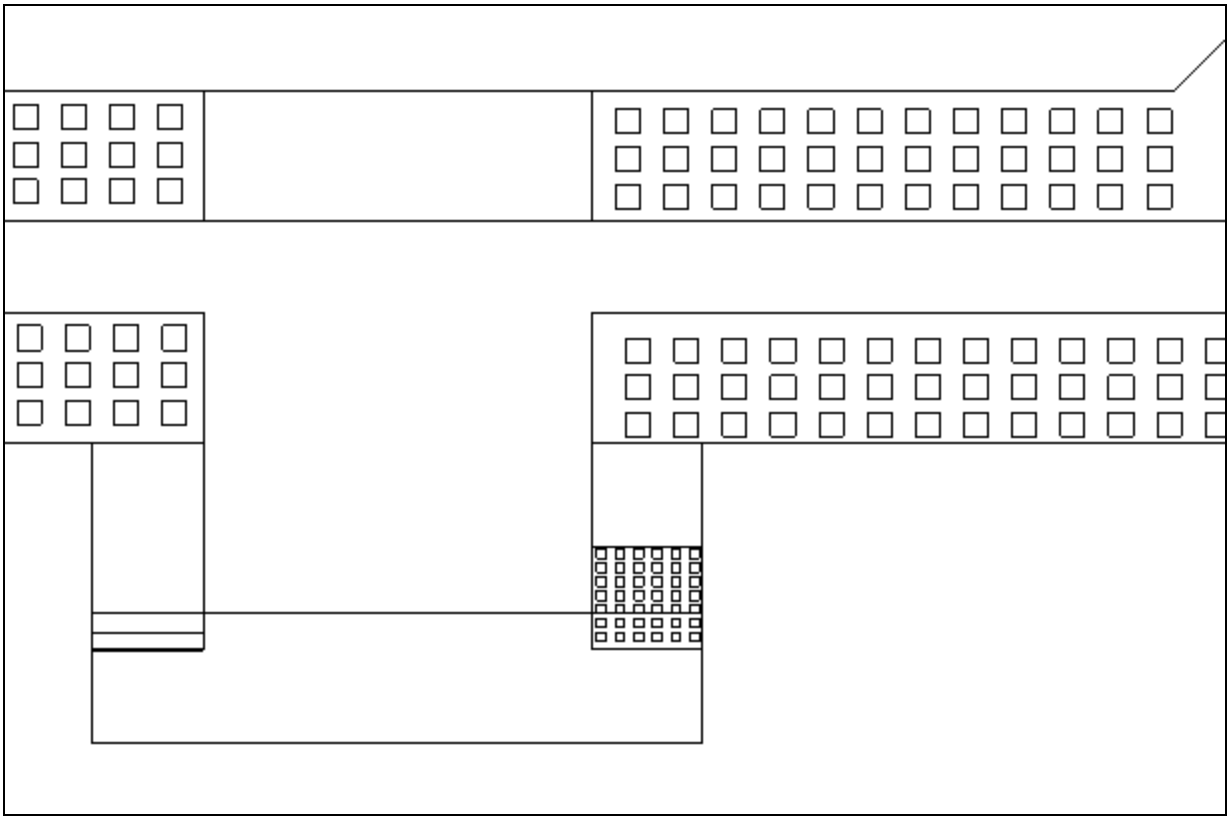


3. From the **Layout** tab, click **Show Physically Connected**.



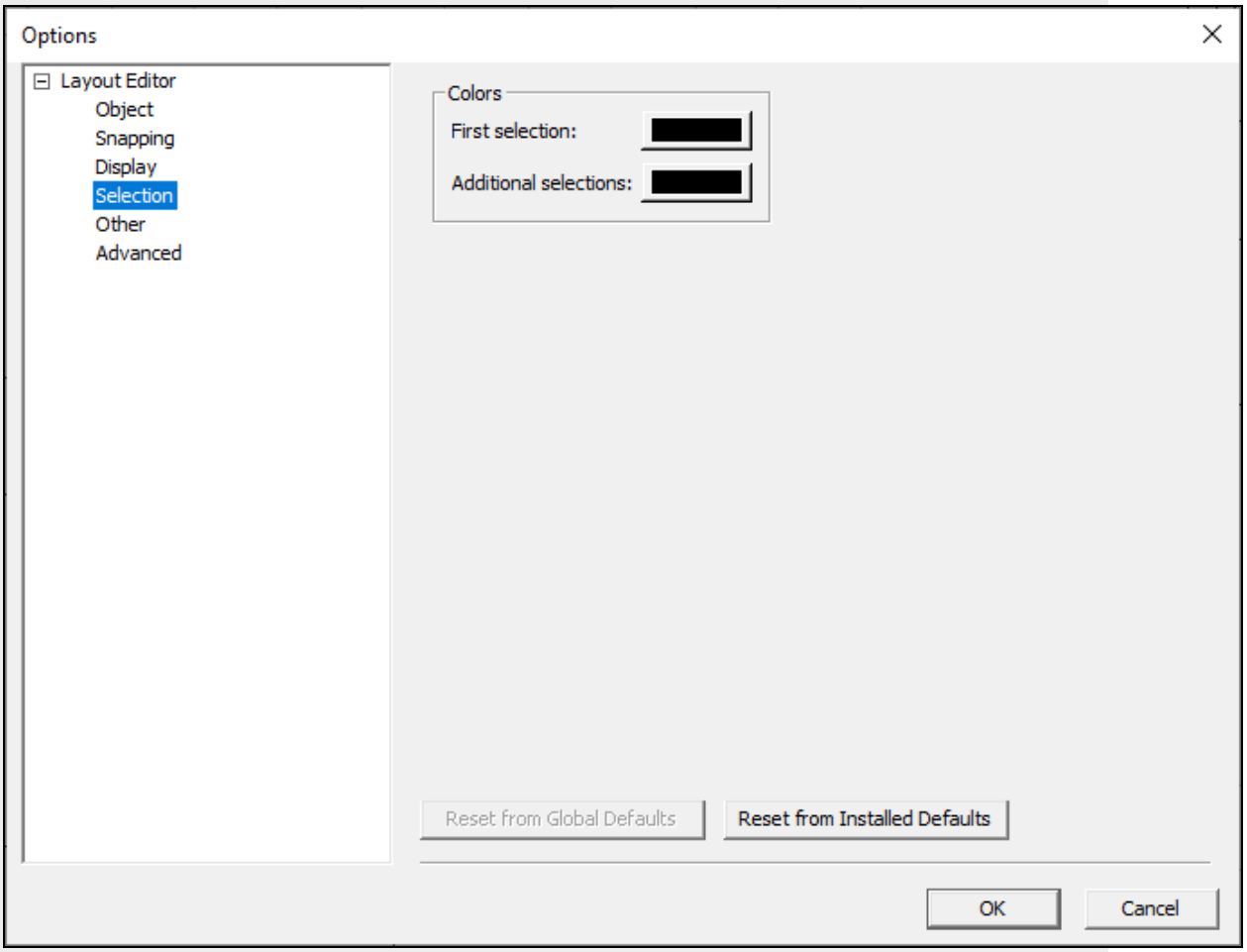
Every element physically connected to the selected element is highlighted.

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.

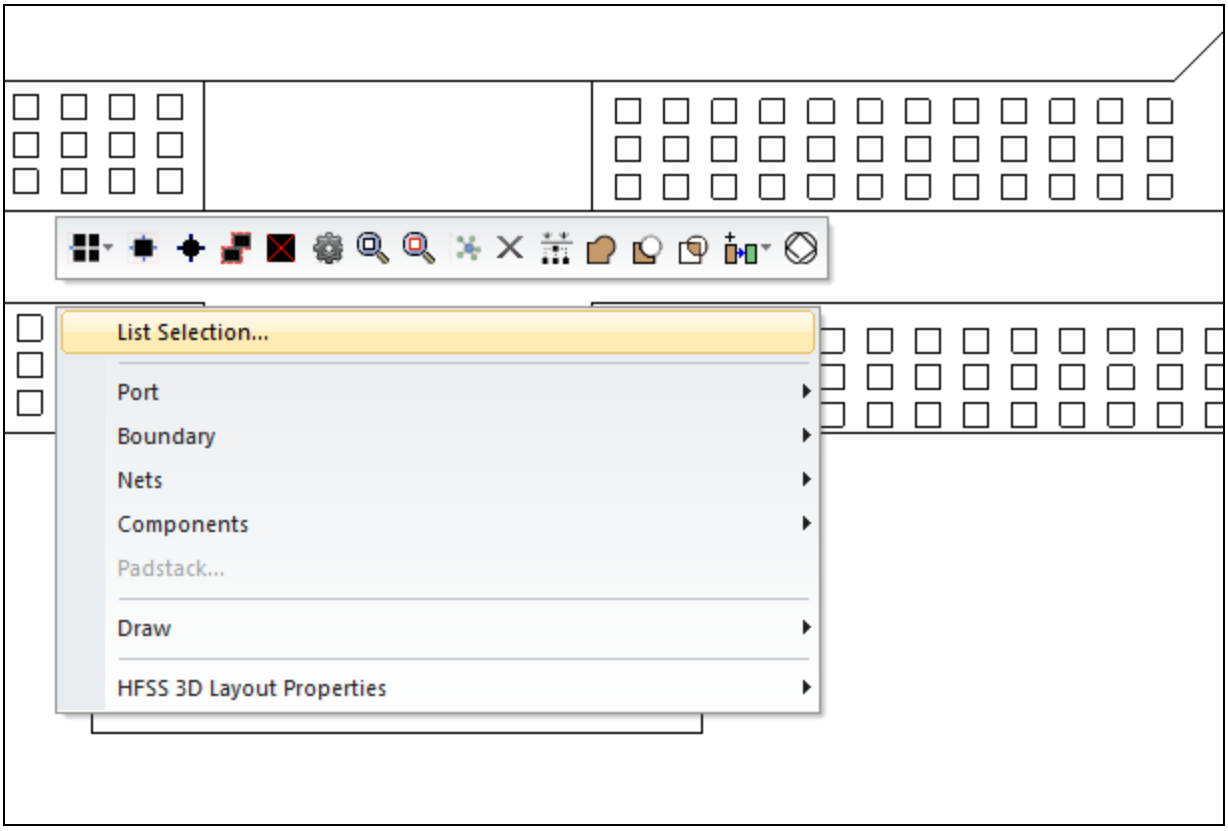
Note: To change the selection and highlight colors, refer to Layout Editor Options: Selection Panel.



Viewing Physically Connected Elements in the Design List Window

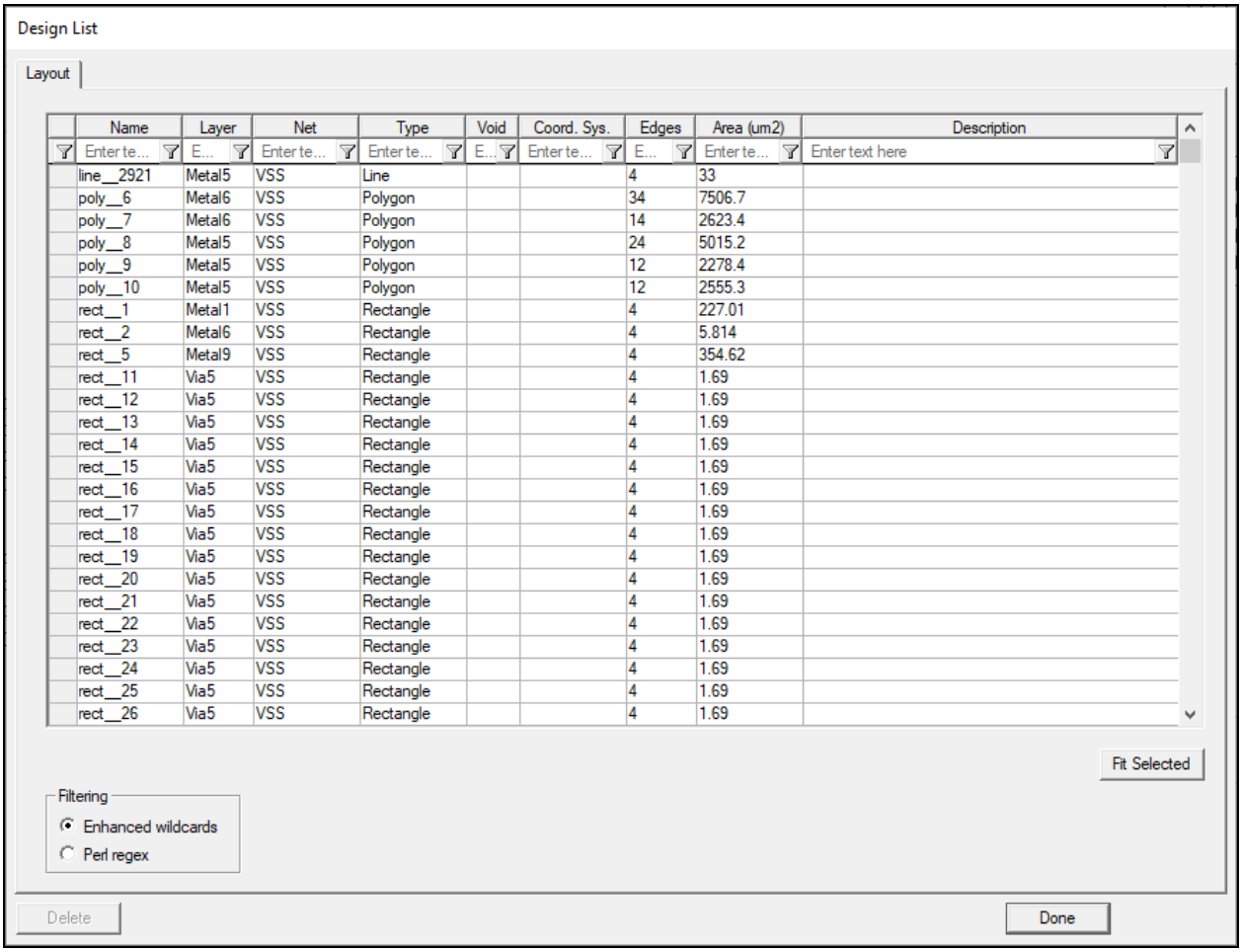
To view a list of the physically connected objects selected in Viewing Physically Connected Elements in IC Mode, right-click within the **Layout Editor** while the selection is still active.

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



Then select **List Selection** to open the **Design List** window.

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



Importing Tabular Arrays

Array data in tabular format can be imported from a file using the **Array Import** wizard. From the **Tools** menu, select **Import Array from Table** to open an explorer window. Then choose *.csv* file to import data from.

The tabular array .csv file should contain the following formatting:

```
arrIndex1    arrIndex2    arrIndex3
c1    "a0"   b1
c2    "a2"   b2
c3    "a3"   b3
```

- First row consisting of list of Array Index Variable names separated by tabs or commas
- Remaining rows consisting of a list of string values separated by tabs or commas

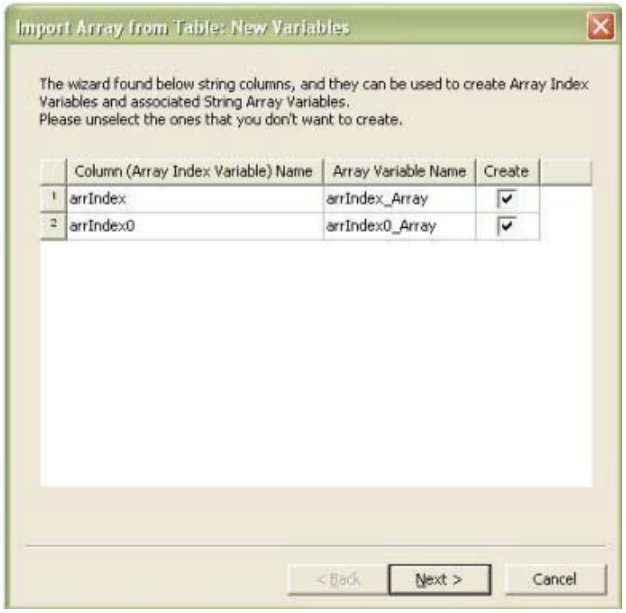
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Double quotation marks (“ “) for string values are optional

After you select a *.csv* file to open.

- If no existing array variables are present in the design, the **Import Array From Table: New Variables** window opens.
- If one or more existing array variables are present in the design, the **Import Array From Table: Existing Variables** window opens.

Each window is shown in the following diagram.



The following controls are available:

- **Column (Array Index Variable) Name** cannot be modified and is used to create the array index variable.
- The associated **Array Variable Name** has the default name “<*ColumnName*>_Array". You can modify the **Array Variable Name**, but variable uniqueness and validity is checked upon closing the wizard.
- Click the **Create** check box to specify that the array index variable should be created.
- Click the control buttons at bottom to go **Back** to the previous wizard window, proceed to the **Next** wizard window, or click **Cancel** to close the window without making changes.

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

If one or more existing array variables are present in the design, clicking **Next** in wizard window opens the **Import Array From Table: Existing Variables** window.

The following controls are available:

- **Array Index Variable** name can *not* be modified.
- **Associated Array Variable** can be modified, but variable uniqueness and validity is checked upon closing the wizard.
- The following choices appear in the **Action** drop-down menu:

  — **Merge with existing value**

  — **Replace existing value**

  — **No Action**

  If you select **Replace existing value**, you are then prompted for verification.

- If you import a file to a design where a string column already exists as an **ArrayIndexVariable**, the value of the variable is checked when imported. If the existing array value is a superset of new array values, the array variable are appear in the **ExistingVariables** window.

Also use the same .csv tabular array file to setup a parametric analysis. See Optimetrics.

When the parametric analysis is set up, the **ArrayIndexVariable** is searched and its index is used to create the table. Whether the **ArrayIndexVariable** refers to a double array or a string array, you must use the array element on the imported file — rather than the index value itself — to set up the **Parametric from File** analysis.

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

Direct Import Cadence APD Links

Direct Import From a Static Data Source

- *.mcm, *.sip, and *.brd requires a binary installation of the Cadence SPB platform. Specifically, extracta.exe must be available and executable in the local environment.
- *.anx is a static data source that can be exported from within Cadence SPB and transported to the local machine. This mechanism allows you to use APDLinks without a local installation of the Cadence tools.

Importing Metal on Nets Without Names

By default, when importing .mcm, .sip, and .brd files, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform filters out all metal on the nets without name. If this behavior is undesirable you need to create a control file with the exact same name as the file you are looking at with the type changed to .xml and located in the same directory. The file should look like the following diagram. If ImportDummyNet is "true" any metal on the dummy net are imported. AEDT also allows control of the assignment of default values to components with the EnableDefaultComponentValues. Legal values for both of these are "true" or "false".

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">
<ImportOptions ImportDummyNet="true"
EnableDefaultComponentValues="true"/>
</c:Control>
```

Layout Links

Utilizing Layout Links take advantage of a closer integration between the 3D Layout and external data sources. Layout Links permits HFSS to host source-specific translation features and allows for plug-ins that behave as “quasi-native” primitives that also use external data semantics. Ensure APDLinks is activated by completing these steps.

1. Select **Tools** > **Layout Links**
2. Click the **Config** tab
3. If applicable, click **AddLink** , then select <installdir>/apdlinks.dll to activate APDLinks.

- Activation of a Link is only required once. On subsequent Ansys Desktop sessions, the link remains available and activate. Once a link is loaded, it is listed in the **AvailableLinks** column of the **Layout Links** tab.
- The Cadence APD/Allegro Link provides **Action** for **Server** configuration, **Options** setup, and **Bondwire** management.

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

Cadence APD Links

Utilizing Cadence APD, take advantage of a closer integration between 3D Layout and external data sources. The following subsections describe Cadence APD link actions regarding servers, options, and bondwires.

Configure Ports for Dynamic Interaction with the Cadence Data Source

- HFSS can interact dynamically with an active Cadence session using IPC. Multiple clients can be configured and ports specified on the **Cadence APD Links** window.

Configure Import Options

- Purge conflicting net geometry: upon import, if the target layout already contains geometry on a given net, delete it.
- Import diestacks: create and adjust layers and geometry using the SiP diestack information.
- Configure discrete components by configuring ports and creating local references for surface mount devices (SMDs).
- Post-import callback by configuring a script to execute after design import.
- Implement naming conventions by specifying how to name imported pins.

Configure and View Active Bondwire Definitions

- Cadence bondwire profile definitions can be loaded and used within HFSS Layout. Profiles found in *.*abwlb* (Ansys Bondwire Definition) files in syslib folder are automatically loaded.
- Profiles are defined and parsed using the Cadence XML Wirebond Document Type Definition.
- Profile definitions can be viewed, imported, and edited in the **Bondwires** tab of the **Edit Libraries** window (i.e., from **Tools** select **Edit libraries** > **Bondwires** to open the **Edit Libraries** window.

SPB License For Direct Import

Utilizing Layout Links and Cadence APD, take advantage of a closer integration between HFSS 3D Layout and external data sources. There are several modes of operations regarding Direct Import, some require an SPB License while some do not.

Prerequisites

The following section details the minimal installations required to implement the various modes of operation for direct import. These operations allow you to take advantage of a closer

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

integration between the 3D Layout and external data sources.

Direct Import of brd/mcm/sip Files

Extracta is an unlicensed utility provided by Cadence. Typically, however, a minimal Cadence installation is required to execute this application. For this method, extracta.exe must be locally executable.

Export/Import

This is similar to the ANF data-transfer model. You can export an *.anx file from a different machine as a standalone file and import it to leverage all the Layout Links features.

Client/Server Interaction

For this, you need to be working in the APD environment, so a Cadence installation and license are required. This also requires Layout Links integration be installed and configured for the SPB environment. Currently, this must be downloaded and manually configured.

Running AnsTranslator on the Command Line

The format for running **AnsTranslator** on the DOS command line is:

```
AnsTranslator <input> <output> <switches>
```

Where:

- `AnsTranslator` is the complete path to AnsTranslator.exe
- `<input>` is the required path to the design to be translated
- `<output>` is the optional path to the edb directory. By default, the export type is edb. If no `<output>` is specified, the edb design is created in the same folder as the `<input>` file/folder.
- `<switches>` are any of the following options:
  - `-i=inputFormat`

    Specifies the file format for translator input. If not specified, the file type is deduced from the file extension. Gerber designs must be explicitly specified though this option.

    Options: AutoCAD, GDSII, Extracta, IPC2581, ODB++, Gerber, or anfv2

  - `-o=outputFormat`

    Specifies the output format. The default output is edb.

    Options: edb and ControlFile

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

> **Note:**
>
> An IPC2581 rev.c exporter has been introduced as a beta feature in Anstranslator. To use this functionality, enter `[design file].aedb [IPC-2581 file].xml -i=edb -o=ipc2581` at the command line. Flex and multi-zone designs, non-laminate designs, and backdrills are not currently supported, and padstack instances that include backdrills will be exported without them. Designs are flattened during export. Circular, square, and rectangular pads are exported as intended, while all other pad shapes are converted to polygons, except thermal pads, which are ignored and not exported. Circular and square holes in the design's padstack definition are exported as intended, whereas all other hole shapes are converted to polygons and exported as a rev. 4 *.xml.

- `-c=controlFileName`

  Specifies the path to a control file (*.xml). If not specified, the control file is still used (*.xml) as long as it has the same name and path as the `<input>` file/folder.

  > **Note:**
  >
  > To generate nets during import, enter:
  >
  > `<GDS_CELL_RULES>`
  >
  > `<WHITE_BOX_CELLS>*</WHITE_BOX_CELLS>`
  >
  > `</GDS_CELL_RULES>`
  >
  > This will automatically generate all nets.

- `-l=logFilename`

  Specifies the path to log file (*.log). By default, the log file is created in the same folder as the `<output>` edb file.

- `-w=workingDirectoryName`

  Specifies the directory to use for any file manipulation needed by the translation. If not specified, the directory for the input is used as the working directory.

- `-t=technologyFile`

  Specifies the path to a technology that defines a stack up. The file can be *.layermap, *.ircx, *.vlc.tech, or *.itf. *.itf must have a mapping file specified with

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

them. Use this switch when a technology file is provided. Users should specify either a control file (-c) **or** a GDS layer mapping file (-g).

- `-f=filterfile.text`

  Use to specify all signal layers to import into the design as a list of layer names separated by end of lines. Via layers will only be imported if both the start and end layers are included in the list. The file can be *.ircx, *.itf, or *.vlc.tech. Requires a technology file (-t) switch.

- `-g=gdsMappingFile`

  Specifies the path to a GDS layer mapping file (*.map).

- `-v`

  Specifies verbose logging (without this option, only warnings and errors are logged).

- `-a(nfcmp)`

  Optionally used for ANFV2 import to specify a *.cmp file

> **Note:**
>
> A new OASIS translator (i.e., Open Artwork System Interchange Standard) has been introduced as a beta feature in Anstranslator. To use this functionality, enter `-i=oasis` at the command line. All arguments that are valid for GDSII files should work with OASIS (*.oasis) files. The **GDSII Import** window in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform now supports importing OASIS files.

Net Importation in AnsTranslator

When bringing in ODB++ and Extracta designs through AnsTranslator, Nets is automatically assigned into two distinct Net/Classes: "Power/Ground Nets" or "Non Power/Ground Nets". These can be overridden through the .xml control file. A sample xml file is included in the following diagram, with an explanation that follows.

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">
 <Nets>
  <Net Name="GND" IsSignalNet="false" />
  <Net Name="VCC"/>
  <Net Name="Battery"/>
  <Net Name="Batt_Sense"/>
```


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

```
   <Net Name="change"/>
   <Net Name="CLK"/>
   <Net Name="data[0]">
      <NetClass Name="data" />
      <NetClass Name="sig" />
    </Net>
   <Net Name="data[1]">
      <NetClass Name="data" />
      <NetClass Name="sig" />
    </Net>
   <Net Name="data[2]">
      <NetClass Name="data" />
      <NetClass Name="sig" />
    </Net>
   <Net Name="data[3]">
      <NetClass Name="data" />
      <NetClass Name="sig" />
    </Net>
   <Net Name="data[4]">
      <NetClass Name="data" />
      <NetClass Name="sig" />
    </Net>
 </Nets>
</c:Control>
```

1. If a net is not included in the list of nets it are not imported into the design. This an excellent way to look at a small subset of the design.
2. If "IsSignalNet" = "false", it is automatically assigned to the "Power/Ground Nets". If it is "true", it are assigned to "Non Power/Ground Nets".
3. If a NetClass has been assigned, it is not placed into the default categories, unless the "IsSignalNet" attribute is set or the NetClass is explicitly specified.

Layout and Control File

Many Anstranslator directories do not contain information that is required by the Ansys Electronics Desktop (AEDT)™ electronics systems design platform and SIwave, such as material and layer characteristics (e.g., height). This information is sometimes exported by the original source program (e.g., Cadence, Mentor Graphics) to the Anstranslator directories (e.g., ODB++ directories) and can then be used by the translator. But if this information is not present

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 directory, or if the information is present but needs to be changed, a separate input control file can be provided by the translator.

AEDT input control file is in XML and can be created using standard third party text editors such as Notepad in Microsoft Windows or VI in Linux. The associated XML schema file, "stackup.xsd," is located in the parent AEDT installation directory. The schema file can be used with third-party editors or scripting tools that check XML syntax. Specific keywords that are valid to use in the control file can be found in the "stackup.xsd" schema. The associated XML schema for the import options information is "ImportOptions.xsd".

You are not required to use a control file when importing translator files. However, if you do not use a control file, stackup and material definitions may not be defined. If you do use a control file, you can adjust the stackup either manually or by importing the control file into the design using the stackup window. If you import the control file, the stackup are modified according to rules described in Importing a Stackup.

Control files may be used for:

- Translator import
- Import using the stackup window
- AutoCAD import — AutoCAD import uses any specified stackup, material, or import option information specified in the control file to fill the import window and set up for import.
- GDSII import — GDSII import uses any specified stackup or material information specified in the control file to fill the import window and set up for import.

An example of a control file follows. The first three lines are heading lines that are required in all control files used with the translator. From the following example, after the heading lines, two other sections are present: one for materials and one for layers.

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">
<Stackup schemaVersion="1.0">
  <Materials>
    <Material Name="FR4_epoxy">
      <Permittivity>
                <Double>4.4</Double>
      </Permittivity>
      <DielectricLossTangent>
                <Double>0.02</Double>
      </DielectricLossTangent>
      <ThermalConductivity>
                <Double>0.294</Double>
```


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

```
    </ThermalConductivity>
    <ThermalExpansionCoefficient>
                <Double>1.5e-05</Double>
    </ThermalExpansionCoefficient>
    <MassDensity>
                <Double>1900</Double>
    </MassDensity>
    <YoungsModulus>
                <Double>11000000000</Double>
    </YoungsModulus>
    <PoissonsRatio>
                <Double>0.28</Double>
    </PoissonsRatio>
  </Material>
  <Material Name="copper">
    <Permeability>
                <Double>0.999991</Double>
    </Permeability>
    <Conductivity>
                <Double>58000000</Double>
    </Conductivity>
    <ThermalConductivity>
                <Double>400</Double>
    </ThermalConductivity>
    <ThermalExpansionCoefficient>
                <Double>1.77e-05</Double>
    </ThermalExpansionCoefficient>
    <MassDensity>
                <Double>8933</Double>
    </MassDensity>
    <YoungsModulus>
                <Double>120000000000</Double>
    </YoungsModulus>
    <PoissonsRatio>
                <Double>0.38</Double>
    </PoissonsRatio>
  </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>
  <ELayers LengthUnit="mm">
    <Dielectrics>
              <Layer Material="air" Name="Air Top Generated"
      Thickness="1.71453000000002e-05"/>
              <Layer Color="#804000" Material="FR4_epoxy"
      Name="Dielectric_1" Thickness="0.1111253"/>
              <Layer Color="#ffff00" Material="FR4_epoxy"
      Name="Dielectric_2" Thickness="0.1695453"/>
              <Layer Color="#800000" Material="FR4_epoxy"
      Name="Dielectric_3" Thickness="0.1619253"/>
              <Layer Color="#004080" Material="FR4_epoxy"
      Name="Dielectric_4" Thickness="0.1695453"/>
              <Layer Color="#ff7a7a" Material="FR4_epoxy"
      Name="Dielectric_5" Thickness="0.1619253"/>
              <Layer Color="#808000" Material="FR4_epoxy"
      Name="Dielectric_6" Thickness="0.2711453"/>
              <Layer Color="#0000ff" Material="FR4_epoxy"
      Name="Dielectric_7" Thickness="0.2711453"/>
              <Layer Color="#fa0000" Material="FR4_epoxy"
      Name="Dielectric_8" Thickness="0.2711453"/>
              <Layer Color="#bdbd00" Material="FR4_epoxy"
      Name="Dielectric_9" Thickness="0.2711453"/>
              <Layer Color="#ff0080" Material="FR4_epoxy"
      Name="Dielectric_10" Thickness="0.2711453"/>
              <Layer Color="#00ff00" Material="FR4_epoxy"
      Name="Dielectric_11" Thickness="0.09334530000000001"/>
    </Dielectrics>
    <Layers>
              <Layer Color="#00ff00" Elevation="2.2231383"
      Material="copper" Name="Top_L1" Thickness="0.0171453"
      Type="conductor"/>
              <Layer Color="#8000ff" Elevation="2.112013"
      Material="copper" Name="GND_1_L2" Thickness="0.0171453"
      Type="conductor"/>
              <Layer Color="#008080" Elevation="1.9424677"
      Material="copper" Name="Inner_Layer_1_L3"
      Thickness="0.0171453" Type="conductor"/>
```


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

```
            <Layer Color="#ff8000" Elevation="1.7805424"
        Material="copper" Name="GND_2_L4" Thickness="0.0171453"
        Type="conductor"/>
            <Layer Color="#00ff80" Elevation="1.6109971"
        Material="copper" Name="Inner_Layer_2_L5"
        Thickness="0.0171453" Type="conductor"/>
            <Layer Color="#3dffff" Elevation="1.4490718"
        Material="copper" Name="GND_3_L6" Thickness="0.0171453"
        Type="conductor"/>
            <Layer Color="#bd00bd" Elevation="1.1779265"
        Material="copper" Name="VCC1_L7" Thickness="0.0171453"
        Type="conductor"/>
            <Layer Color="#008000" Elevation="0.9067812"
        Material="copper" Name="VCC2_L8" Thickness="0.0171453"
        Type="conductor"/>
            <Layer Color="#00bdbd" Elevation="0.6356359"
        Material="copper" Name="GND_4_L9" Thickness="0.0171453"
        Type="conductor"/>
            <Layer Color="#408000" Elevation="0.3644906"
        Material="copper" Name="Inner_Layer_3_L10"
        Thickness="0.0171453" Type="conductor"/>
            <Layer Color="#000080"
        Elevation="0.09334530000000001" Material="copper" Name="GND_
        5_L11" Thickness="0.0171453" Type="conductor"/>
            <Layer Color="#804000" Elevation="0"
        Material="copper" Name="Bottom_L12" Thickness="0.0171453"
        Type="conductor"/>
            <Layer Color="#353981" Name="Outline"
        Type="outline"/>
      </Layers>
    </ELayers>
  </Stackup>
</c:Control>
```

Control File Usage with Renaming

Three examples of a GDS control file follow. The first was output by the GDS import window. The second was output on the Layer Stackup window and the third, which is suitable for import, was constructed by making sure the layer mapping is identical in the second file as it was in the first. Note that not all Control File settings are applicable 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.

> **Note:**
>
> The following example was created for GDS import but the principle applies to importing Autocad and ODB++. Layer remapping is not supported for the importing of Cadence APD/Allegro/Sip.

Name="30" refers to the layer in the original design.

TargetLayer="Top" refers to the new name of the layer.

1. From **Import** window

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>

 <c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">

<Stackup schemaVersion="1.0">

<Materials>

[[[[[[[[[[[[[ Deleted for space reasons ]]]]]]]]]]]]]

</Materials>

<Layers LengthUnit="mm">

<Layer Color="#008000" FillMaterial="FR4_epoxy" GDSIIVia="false"
Material="copper" Name="30" TargetLayer="Top" Thickness="0"
Type="conductor"/>

<Layer Color="#004080" FillMaterial="FR4_epoxy" GDSIIVia="false"
Material="copper" Name="55" TargetLayer="Bottom" Thickness="0"
Type="conductor"/>

</Layers>

</Stackup>

 <ImportOptions Flatten="true" GDSIIConvertPolygonToCircles="false"/>

</c:Control>
```

If the Ansys Electronics Desktop (AEDT)™ electronics systems design platform exports on the Layer Stackup window, note the layer that is called "30" in the gds design is referred to as "Top" in the example. To use the example file, make the substitution Name="Top" with Name ="30" TargetLayer="Top", etc. You are able to define your dielectric layers, etc., in AEDT and use the modified file for repeated translation.

2. From **Layer Stackup** window

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

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>

 <c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">

<Stackup schemaVersion="1.0">

<Materials>

[[[[ Deleted again]]]]

</Materials>

<ELayers LengthUnit="mm">

<Dielectrics>

<Layer Color="#008000" Material="FR4_epoxy" Name="Dielectric"
Thickness="10"/>

</Dielectrics>

<Layers>

<Layer Color="#004080" Elevation="5" Material="copper" Name"55"
LayerTarget="bottom" Thickness="5" Type="conductor"/>

<Layer Color="#008000" Elevation="0" Material="copper" Name="30"
LayerTarget="top" Thickness="5" Type="conductor"/>

</Layers>

</ELayers>

 </Stackup>

</c:Control>
```

3. From Layer Stackup window preserving original name mapping.

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>

 <c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">

<Stackup schemaVersion="1.0">

<Materials>

[[[[ Deleted again]]]]

</Materials>

<ELayers LengthUnit="mm">
```


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

```
<Dielectrics>

<Layer Color="#008000" Material="FR4_epoxy" Name="Dielectric"
Thickness="10"/>

</Dielectrics>

<Layers>

<Layer Color="#004080" Elevation="5" Material="copper" Name="bottom"
Thickness="5" Type="conductor"/>

<Layer Color="#008000" Elevation="0" Material="copper" Name="top"
Thickness="5" Type="conductor"/>

</Layers>

</ELayers>

 </Stackup>

</c:Control>
```

Using Excel to Generate a Control File

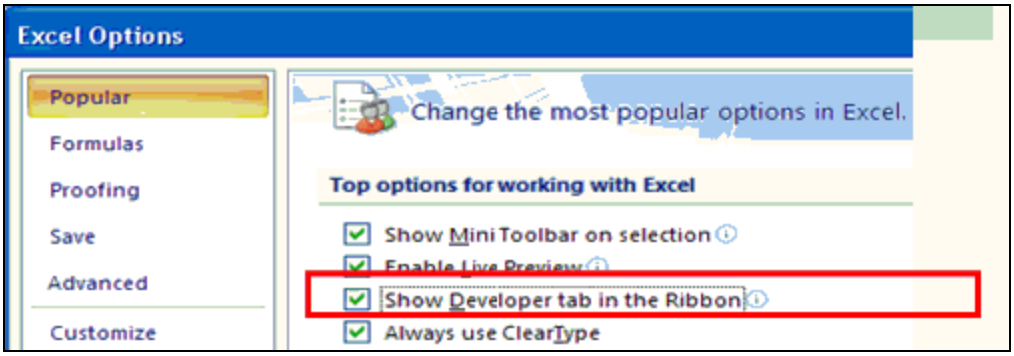
It is possible to use XML schema functionality in Microsoft Excel to create stackup-layer information that can then be exported to an ODB++ control file.

An XML schema is basically a set of rules (constraints) that define the structure and content (elements) of an XML file. Both XML and XML Schema documents (*.xml, *.xsd) are straightforward user-friendly text files, and Microsoft Excel offers an intuitive interface that “maps” elements in an XML schema to data cells in an ordinary worksheet. As such, and because the XML schema is quite simple, it is possible to use a plain-text XML editor in Microsoft Excel to create stackup-layer controls that are suitable for Ansys tools. These controls can then be exported to an Control File.

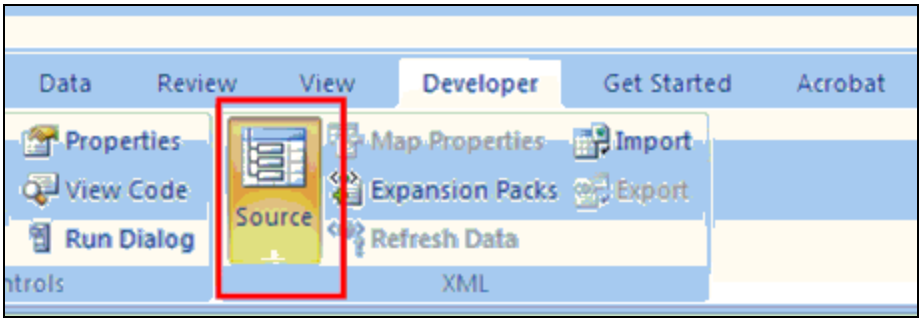
The schema used in the following example is shipped with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform v6.0 (or higher) and can be found at <*desktop_inst_dir*>\Stackup.xsd, where <*desktop_inst_dir*> is the AEDT installation directory. In this example, Microsoft Excel 2007 is used, but Excel 2003 should also be compatible.

1. **Access XML Functionality in Excel:** To access the XML import/export functionality in Excel, you must first activate the Developer Tab. If the Developer Tab is not already visible, click the Office Button , then select **Excel Options**.

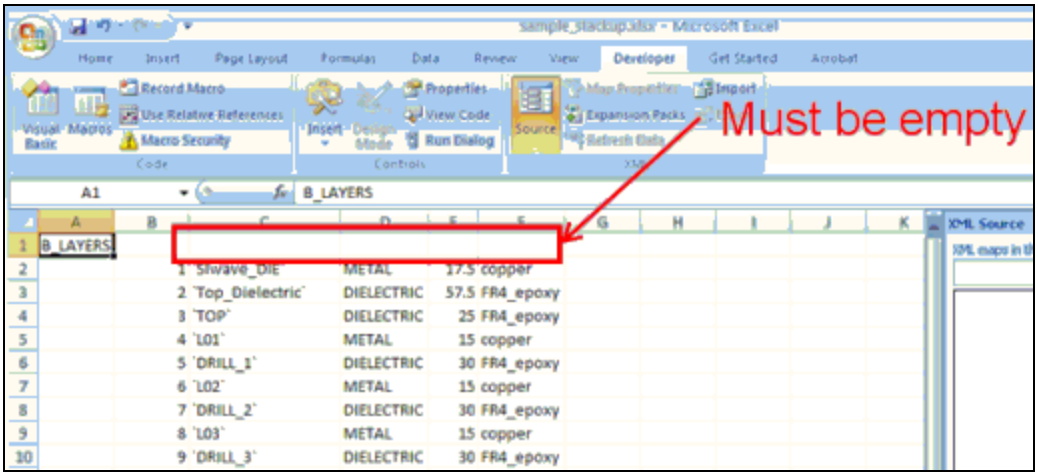
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If applicable, select the **Popular** category, then click **Show Developer tab in the Ribbon**. Then, on the Developer tab in the XML group, click **Source** to display the XML Source task-pane.

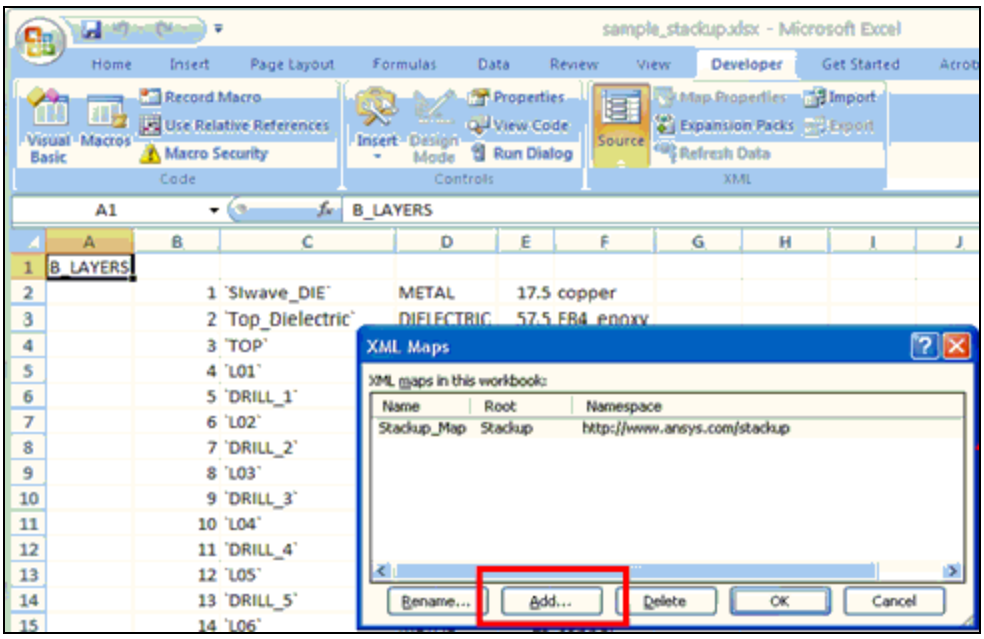


3. **Open the Excel Spreadsheet:** Open an appropriate spreadsheet file (*.xls or *.xlsx) to export to the control file.

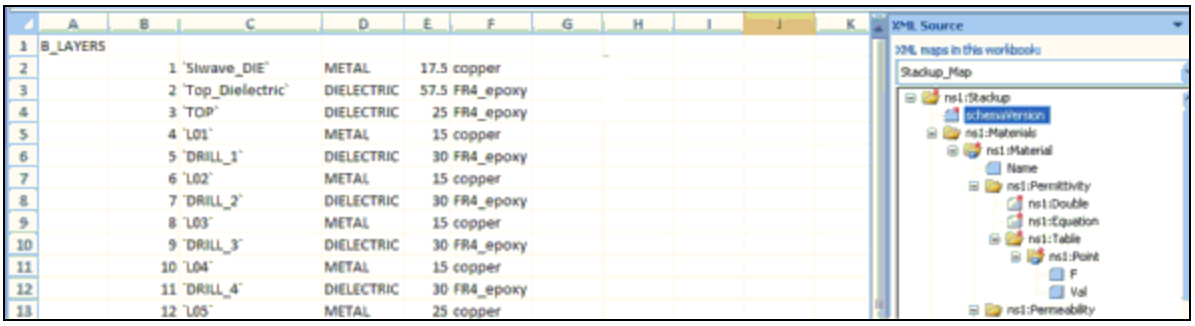


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. **Load the Ansys XML Schema:** In the XML Source task pane, click XML Maps to open the XML Maps window box.



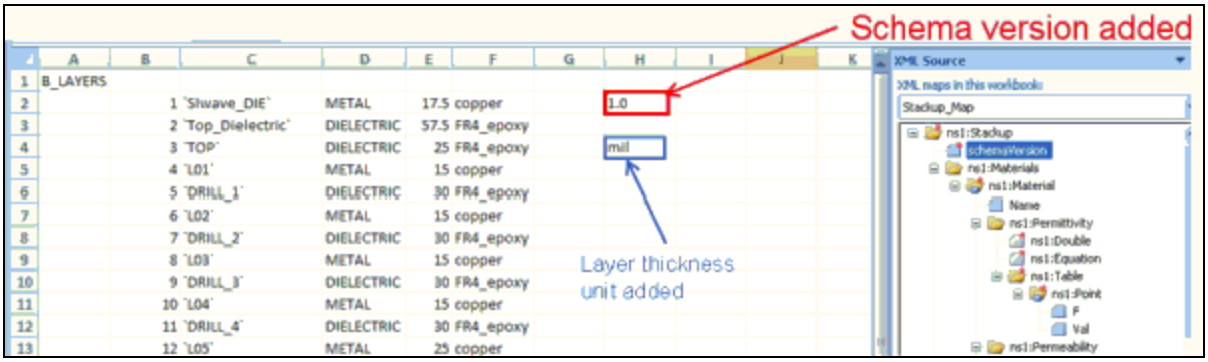
5. **Load the Ansys XML Schema** (contd)**:** Click **Add** to open the Select XML Source window, then browse to the .xsd file in the AEDT installation directory and click **Open.** Next, click **OK** to close the Source window. The XML schema are loaded into the XML Source Pane.



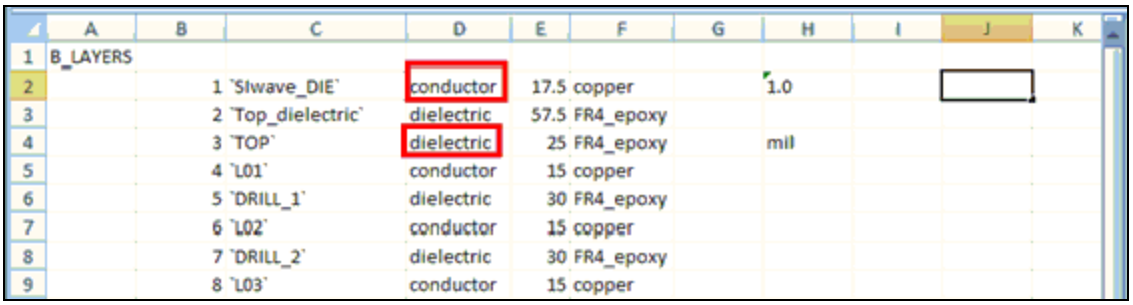
6. **Prepare the Excel Spreadsheet**: From the schema element list that appears in the task pane, those elements with a red star are mandatory fields that you must map to cells in the worksheet. But, you do not need to map a starred element if its parent is not also starred. For most Excel files, you need to add at least one cell in the worksheet that contains the schema version number — which is currently fixed at “1.0” — you can add it anywhere, but it must be formatted as “text”, instead of the default format “number”, so that the literal string are exported. You may add additional fields to the worksheet, such as a layer-

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

thickness unit, but this is not required.



7. **Prepare the Excel Spreadsheet** (contd): Many pre-existing layer-stackup files may contain names/symbols that are different from those used by the XML Schema. These names are flagged as errors, even if they represent the same entity. Because of this, these names must be replaced with valid entries before being exported to XML (e.g., “METAL” is not a valid layer type and is replaced with “conductor”, and since XML is case-sensitive, “DIELECTRIC” is replaced with “dielectric”).



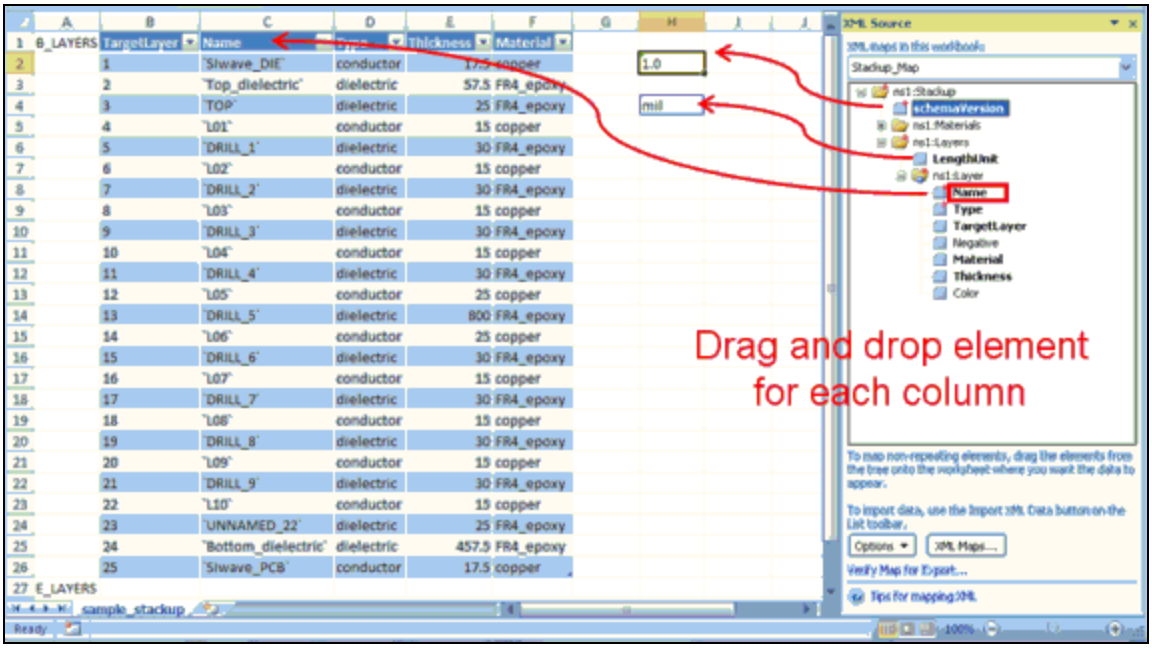
	A	B	C	D	E	F	G	H	I	J	K
1	B_LAYERS										
2		1	`Slwave_DIE`	conductor	17.5	copper		1.0			
3		2	`Top_dielectric`	dielectric	57.5	FR4_epoxy					
4		3	`TOP`	dielectric	25	FR4_epoxy		mil			
5		4	`L01`	conductor	15	copper					
6		5	`DRILL_1`	dielectric	30	FR4_epoxy					
7		6	`L02`	conductor	15	copper					
8		7	`DRILL_2`	dielectric	30	FR4_epoxy					
9		8	`L03`	conductor	15	copper					

For a complete list of valid entries for each element, open the XML Schema document (*.xsd) in a text editor and search for the element name.

8. **Map the XML schema to the workbook**: Drag each element on the task pane to the corresponding cell in the workbook. The mapped areas appear in the blue nonprintable borders. Repeat this mapping operation for each element by dragging it into the workbook.

- For columns of data, drop the element on the cell directly above the top row in order to make it the header.
- For a single cell, drop the element on the cell itself.

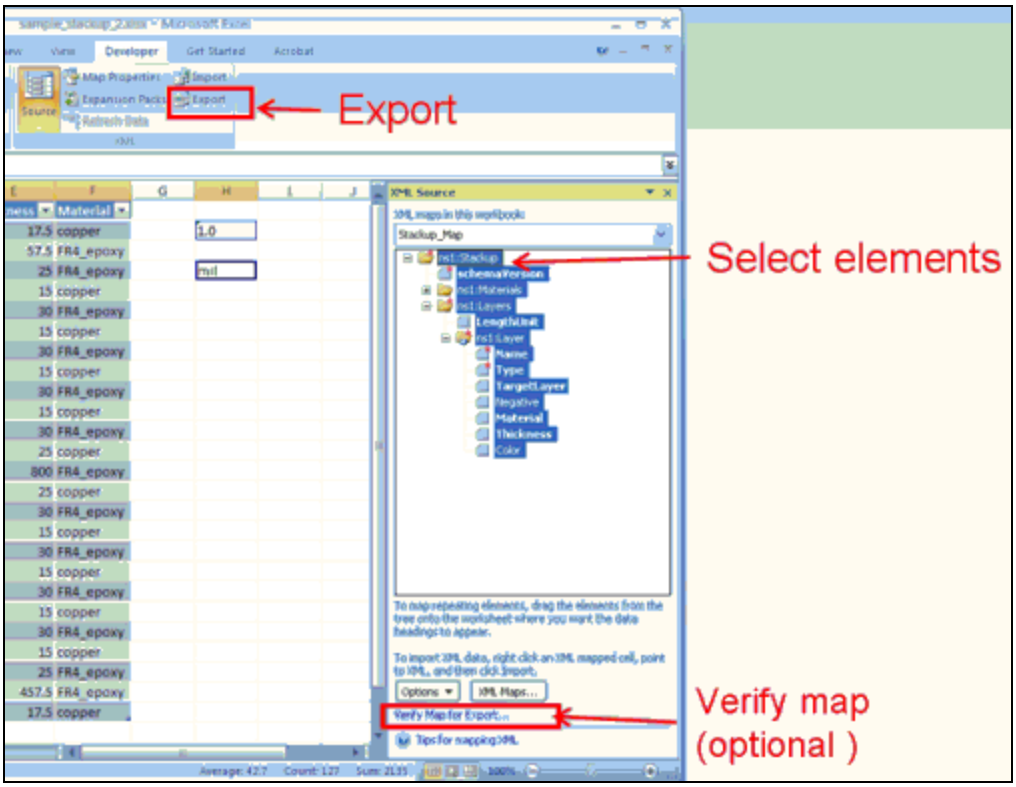
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. **Export to XML File:** Select elements in the task pane to export them. All corresponding mapped cells are highlighted in the worksheet (Hint: To select all, click on **ns1::Stackup**). From the **Developer** tab, click **Export**. From the Export XML window that opens, specify a

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

name for the file. Next, click **Save** and the file is exported to XML.



10. **Review the XML File:** You can open the exported XML file in a text editor or Internet Explorer for review.

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

```
<?xml version="1.0" encoding="UTF-8" standalone="yes" ?>
<ns1:Stackup xmlns:ns1="http://www.ansys.com/stackup" schemaVersion="1">
- <Layers LengthUnit="mil">
    <Layer Name="`SIwave_DIE`" Type="conductor" TargetLayer="1" Material="copper" Thickness="17.5" />
    <Layer Name="`Top_dielectric`" Type="dielectric" TargetLayer="2" Material="FR4_epoxy" Thickness="57.5" />
    <Layer Name="`TOP`" Type="dielectric" TargetLayer="3" Material="FR4_epoxy" Thickness="25" />
    <Layer Name="`L01`" Type="conductor" TargetLayer="4" Material="copper" Thickness="15" />
    <Layer Name="`DRILL_1`" Type="dielectric" TargetLayer="5" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L02`" Type="conductor" TargetLayer="6" Material="copper" Thickness="15" />
    <Layer Name="`DRILL_2`" Type="dielectric" TargetLayer="7" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L03`" Type="conductor" TargetLayer="8" Material="copper" Thickness="15" />
    <Layer Name="`DRILL_3`" Type="dielectric" TargetLayer="9" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L04`" Type="conductor" TargetLayer="10" Material="copper" Thickness="15" />
    <Layer Name="`DRILL_4`" Type="dielectric" TargetLayer="11" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L05`" Type="conductor" TargetLayer="12" Material="copper" Thickness="25" />
    <Layer Name="`DRILL_5`" Type="dielectric" TargetLayer="13" Material="FR4_epoxy" Thickness="800" />
    <Layer Name="`L06`" Type="conductor" TargetLayer="14" Material="copper" Thickness="25" />
    <Layer Name="`DRILL_6`" Type="dielectric" TargetLayer="15" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L07`" Type="conductor" TargetLayer="16" Material="copper" Thickness="15" />
    <Layer Name="`DRILL_7`" Type="dielectric" TargetLayer="17" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L08`" Type="conductor" TargetLayer="18" Material="copper" Thickness="15" />
    <Layer Name="`DRILL_8`" Type="dielectric" TargetLayer="19" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L09`" Type="conductor" TargetLayer="20" Material="copper" Thickness="15" />
    <Layer Name="`DRILL_9`" Type="dielectric" TargetLayer="21" Material="FR4_epoxy" Thickness="30" />
    <Layer Name="`L10`" Type="conductor" TargetLayer="22" Material="copper" Thickness="15" />
    <Layer Name="`UNNAMED_22`" Type="dielectric" TargetLayer="23" Material="FR4_epoxy" Thickness="25" />
    <Layer Name="`Bottom_dielectric`" Type="dielectric" TargetLayer="24" Material="FR4_epoxy" Thickness="457.5" />
    <Layer Name="`SIwave_PCB`" Type="conductor" TargetLayer="25" Material="copper" Thickness="17.5" />
  </Layers>
</ns1:Stackup>
```

Exporter Issues

The following known issues have been found regarding translated directories. The problems occur when the contents of a file do not completely conform to the Valor specifications. Anstranslator response varies depending on the nature of the problem, and an informative warning/error message is provided.

Extra text after the part-name of a CMP record in a components file

The one-word component part-name should be the final text before the terminating semi-colon in a CMP record (";"). Any extra text is ignored.

Example with the extra text in bold:

```
CMP 8 0.94074 0.3248 270.0 N J1 mPCIE EDGECONN ;0=1,1=0.000010
```

Duplicate pin numbers in TOP records in a components file

Each toeprint record (TOP) for a component should have a distinct pin number. When a duplicate is found, the duplicate pin number is changed to an unused number. Users are given a warning.

Example:

The bold pin numbers are duplicates.

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

...

```
TOP 14 -0.1284500 0.1181400 90.0000000 N 108 0 15
TOP 15 0.1284500 0.1181400 90.0000000 N 85 0 16
TOP 16 -0.1284500 0.1575100 90.0000000 N 109 0 17
TOP 17 0.1284500 0.1575100 90.0000000 N 7 0 18
TOP 18 0.0000000 0.3960000 90.0000000 N 1 0 G1
TOP 18 – 0.1140000 0.2820000 90.0000000 N 1 0 G1
TOP 18 0.0790000 0.3530000 90.0000000 N 1 0 G1
TOP 18 – 0.0790000 0.3530000 90.0000000 N 1 0 G1
TOP 18 0.1140000 0.2820000 90.0000000 N 1 0 G1
TOP 19 0.0000000 -0.3700000 90.0000000 N 1 0 G2
TOP 19 – 0.1140000 -0.2820000 90.0000000 N 1 0 G2
TOP 19 0.1140000 -0.2820000 90.0000000 N 1 0 G2
TOP 19 0.0790000 -0.3530000 90.0000000 N 1 0 G2
TOP 19 – 0.0790000 -0.3530000 90.0000000 N 1 0 G2
```

A NET record in an eda data file contains extra text

The net name specified in the NET line of an eda data file should not contain spaces. If spaces are present, they are replaced with underscores. A warning is issued. This situation could happen when net names with spaces are exported without removing the spaces.

Example:

These NET lines from an edata/data file result in nets named JTG_ENA_TOP and JTG_ENA_PHY.

```
#NET 973
NET JTG_ENA TOP
...
#NET 974
NET JTG_ENA PHY
```

Duplicate pin names

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

Duplicate pin names in the same component — as defined by a PKG record in an eda data file. When a pin name is duplicated, the translator forces it to be unique by appending a number. A warning is issued.

Example with pin names 11 through 18 used twice in the PKG record:

```
# PKG 2
PKG smnhda18.2_39_ni 0.03937 -0.4084999 -0.24 0.3825 0.24
RC -0.251 -0.24 0.502 0.48
PIN 1 S 0.15745 -0.1284499 0 U U
RC 0.14545 -0.1634499 0.024 0.07
PIN 2 S 0.15745 0.12845 0 U U
RC 0.14545 0.09345 0.024 0.07
PIN 3 S 0.11808 -0.1284499 0 U U
RC 0.10608 -0.1634499 0.024 0.07
PIN 4 S 0.11808 0.12845 0 U U
RC 0.10608 0.09345 0.024 0.07
PIN 5 S 0.07871 -0.1284499 0 U U
RC 0.06671 -0.1634499 0.024 0.07
PIN 6 S 0.07871 0.12845 0 U U
RC 0.06671 0.09345 0.024 0.07
PIN 7 S 0.03934 -0.1284499 0 U U
RC 0.02734 -0.1634499 0.024 0.07
PIN 8 S 0.03934 0.12845 0 U U
RC 0.02734 0.09345 0.024 0.07
PIN 9 S -0.0000299 -0.1284499 0 U U
RC -0.0120299 -0.1634499 0.024 0.07
PIN 10 S -0.0000299 0.12845 0 U U
RC -0.0120299 0.09345 0.024 0.07
PIN 11 S -0.0393999 -0.1284499 0 U U
RC -0.0513999 -0.1634499 0.024 0.07
```


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

```
PIN 12 S -0.0393999 0.12845 0 U U
RC -0.0513999 0.09345 0.024 0.07
PIN 13 S -0.0787699 -0.1284499 0 U U
RC -0.0907699 -0.1634499 0.024 0.07
PIN 14 S -0.0787699 0.12845 0 U U
RC -0.0907699 0.09345 0.024 0.07
PIN 15 S -0.1181399 -0.1284499 0 U U
RC -0.1301399 -0.1634499 0.024 0.07
PIN 16 S -0.1181399 0.12845 0 U U
RC -0.1301399 0.09345 0.024 0.07
PIN 17 S -0.1575099 -0.1284499 0 U U
RC -0.1695099 -0.1634499 0.024 0.07
PIN 18 S -0.1575099 0.12845 0 U U
RC -0.1695099 0.09345 0.024 0.07
PIN G1 T -0.3959999 0 0 U U
CR -0.3959999 0 0.0125
PIN G2 T 0.37 0 0 U U
CR 0.37 0 0.0125
PIN 11 S -0.0393999 -0.1284499 0 U U
RC -0.0513999 -0.1634499 0.024 0.07
PIN 12 S -0.0393999 0.12845 0 U U
RC -0.0513999 0.09345 0.024 0.07
PIN 13 S -0.0787699 -0.1284499 0 U U
RC -0.0907699 -0.1634499 0.024 0.07
PIN 14 S -0.0787699 0.12845 0 U U
RC -0.0907699 0.09345 0.024 0.07
PIN 15 S -0.1181399 -0.1284499 0 U U
RC -0.1301399 -0.1634499 0.024 0.07
```


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

```
PIN 16 S -0.1181399 0.12845 0 U U
RC -0.1301399 0.09345 0.024 0.07
PIN 17 S -0.1575099 -0.1284499 0 U U
RC -0.1695099 -0.1634499 0.024 0.07
PIN 18 S -0.1575099 0.12845 0 U U
RC -0.1695099 0.09345 0.024 0.07
```

Property line contains multiple words for property name

A property line (PRP) should contain a single-word property-name followed by a quoted string for the value. Some PRP lines have multiple words that precede the quoted value. In this case, it is assumed that the extra words are intended to be part of the property name and they are then concatenated together with underscores. A warning is issued.

Example:

```
PRP SIMULATION MODEL 'MAX2003'
```

Property line contains extra text after value

A property line (PRP) should contain a single-word property-name followed by a quoted string for the value. Some PRP lines have multiple-quoted values. The text after the first quoted value is ignored. A warning is issued.

Example:

```
PRP PART LABEL 'RELE' TN2C'
```

Property line is missing value

A property line (PRP) should contain a single-word property-name followed by a quoted string for the value. Some PRP lines are missing the quoted value. When this occurs, the property is ignored. A warning is issued.

Example:

```
PRP Manufacturer
```

Discrepancy between component file and eda file

A correlation exists between the toeprints specified in the component file and those specified in the eda file. If the files are created incorrectly and their indexes do not match, the Anstranslator aborts translating the components file. Overall translation is continued, but components, padstacks, and nets are not translated. A warning is issued.

Negative net index specified in components file

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

Toeprints in a component file should have a positive net index. If the net index is negative, the Anstranslator aborts translating the components file. Overall translation is continued with padstacks and nets being translated, but components are not translated. A warning is issued.

Example with negative net index in bold:

```
TOP 1 6.31 7.9018001 0.0 N -1 0 2_3
```

The eda data file contains a trace subnet with a hole feature

Trace subnets (part of eda data file) should have only copper features as members. If holes are found as trace members, the hole feature is ignored. A warning is issued.

Example with hole feature in bold.

```
SNT TRC

FID H 6 4717
```

Features from multiple padstacks are listed in the same toeprint in the eda data file

The features listed for a toeprint in an eda data file can be on the same padstack. When features from multiple padstacks are found in one toeprint warnings are issued. The first padstack is translated normally. Pads and hole on the other padstacks are merged into the first when possible.

Arc definition is incorrect

Arcs are specified in directories with a start point, end point, and center point. The distance on the start point to the center should be the same as the distance on the end point to the center. If the two distances are different, an arc can not be created, and instead, the start and end points are used to create a segment (0 height arc). A warning message is issued.

Holes not specified in eda data file

Padstack via and pin holes are specified in the eda data file of the directories. Following is an excerpt of an eda via specification. FID entries specify each part of a via: holes, pads, and antipads. Entries containing "C" are pads (copper), and entries containing "L" are antipads (laminate). Lines containing "H" specify holes, but there are no holes specified in this example because a known problem exists wherein hole features are not specified in the eda data file.

```
SNT VIA

FID C 2 8479

FID C 3 10992

FID L 4 1310

FID L 5 2414

FID L 8 1310
```


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

```
FID L 11 1310

FID L 14 2053

FID L 15 1310

FID C 16 5566

FID C 17 3399
```

If "H" lines are present they indicate which feature from which layer is to be used for the hole. Hole features are found on ODB drill layers. There is a drill layer defined in this ODB folder and it does have features. But because of a known problem, holes are not specified in the eda data file, so “H” lines are not present in the file.

A line in the feature file contains a non-numeric attribute value

The feature file contains lines that list numeric values which specify the setting of various attributes and options. The following feature line mistakenly contains an attribute with a string value instead of a numeric value.

```
P 3.6765000 2.4700000 0 P 0 0;0=0,1=NO,2=1
```

The attribute are not used and a warning message is issued.

Non-numeric value for attribute on feature line in the features file

Feature lines may have attributes specified at the end of the line. The value for an attribute is omitted for boolean attributes. Otherwise it is an number representing either the numerical value for integer and float attributes, an option number for option attributes, or an integer referencing a string on the attribute text string section of the file. The following feature line has a value that is a string, rather than either being omitted or a number as is correct.

```
P 3.6765000 2.4700000 0 P 0 0;0=0,1=NO,2=1
```

The attribute are not used and a warning message is issued.

Missing package name

The PKG record in the eda data file is missing the required package name. There should be a package name in the underlined space underneath.

```
# PKG 0

PKG 0.1 -0.1275 -0.0275 0.9275 1.0275
```

Package contour hole before island in eda data file

A package contour specification for a footprint outline mistakenly specifies a **hole (H)** before it specifies the **island (I)** that could contain it. The hole is ignored and a warning is issued.

```
# PKG 20
```


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

```
PKG T 0.03937 -0.2411574 -0.0492126 0.1688426 0.146504
CT
OB 0.0327165 0.040748 H
OS -0.068451 0.040748
OS -0.068451 0.040441
OS 0.0327165 0.040441
OS 0.0327165 0.040748
OE
OB -0.068451 0.0856299 I
OS 0.0327165 0.0856299
OS 0.0327165 0.1415828
...
```

Features on negative layers specified to be copper rather than laminate

Subnet toeprint features in the eda data file that are on negative layers should be specified as laminate. Some tools incorrectly export them as copper. When this deviation is supported, the feature is used as if it is laminate; the feature becomes an antipad. Copper features in a subnet toeprint are pads. If the FID line under layer 5 is a negative layer, the "C" should be an "L".

```
SNT TOP B 32 0
FID C 5 598
```

Layer Mapping and XML Control Files

Layer mapping files specify the mapping of layers when importing or exporting data with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. The imported or exported layer name or number is mapped to a layer in the layout. The layers mapped in the file are the one that are imported or exported.

- Opening a layer mapping file from an import or export window sets the window controls with the contained mapping information.
- The **Layout Editor** supports two formats of layer mapping files: .layermap and .tech.
- The .tech format allows additional layer information to be supplied when importing.
- If all conductive layers have thickness, a design's three dimensional connectivity are used to generate nets.

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

Layer mapping files are supported with AutoCAD and GDSII import, which also allows for the use of an XML control file.

- For advanced rules on importing GDSII designs, see GDS Translation using XML Control Files.
- For information on via layers in GDSII control files, see Via Grouping and Snapping in XML Control Files.
- For more information on control files, see Layout and ODB++ Control File.

.layermap File Format

- No comment character
- Each layer information entry is specified by a line that contains:

  <import layer> — Name of the DXF layer or number of the GDSII layer

  <destination layer> — Name of the layout layer in AEDT (i.e., the name to map the imported layer to)

Example .layermap file for DXF import

```
S12 signal12
TR trace
S3 signal3
```

Examples .layermap file for GDSII import

```
12 signal12
35 trace
3 signal3
```

This example file creates two layers from stream 74: "ap" and "Cu_PPI":

```
74 0 ap
74 10 Cu_PPI
```

This example maps items from stream 73 and 74 to the layer "ap":

```
73 10 ap
74 0 ap
```

.tech File Format

- A forward slash, “/”, is the comment character.
- Units may be specified with a “UNITS <string>” entry that precedes the layer information t entries, where “<string>” is any of the allowed desktop length units; the default is “nm”.
- Each layer information entry is specified by a line that contains 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.

<import layer> — Name of the DXF layer or number of the GDSII layer

<destination layer> — Name of the layout layer in AEDT (i.e., the name to map the imported layer to)

<layer color> — A color string choice on the following list via the ***Layer Colors*** link.

<layer elevation> — Double

<layer height> — Double

Example .tech file for DXF import

```
UNITS um
/ ---------------------------------------------------------------
/ import# destination Color Elevation Thickness
/ ---------------------------------------------------------------
S12 signal12 blue 1100 530
TR trace red 6620 530
S3 signal3 yellow 8150 2000
```

Example .tech file for GDSII import

```
UNITS um
/ ---------------------------------------------------------------
/ import# destination Color Elevation Thickness
/ ---------------------------------------------------------------
12 signal12 blue 1100 530
35 trace red 6620 530
3 signal3 yellow 8150 2000
```

Click following ***Layer Colors*** link to display a listing of the supported color types and their corresponding RGB values: Layout Editor Layer Colors

Control File Support

Control file support is provided by adding a new GDSDataType attribute that corresponds to the Data Type in the layer stackup section. Note that while multiple-stream data-attributes can be combined into a single layer, all other elements should be identical or one is selected at random.

The following layer stackup results in vias being constructed between the "new" and "old" layers. Note the GDSIIVia = "true" geometries are removed on the design, but they must still specify TargetLayer = "via" in order to do the initial translation.

```
<Layers LengthUnit="mm">
```


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

```
    <Layer Color="#008000" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="new" GDSIIVia="false" Material="copper" Name="30"
    Thickness="10" Type="conductor"/>
    <Layer Color="#008000" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="new" GDSIIVia="false" Material="copper" Name="31"
    Thickness="10" Type="conductor"/>
    <Layer Color="#bd00bd" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="via" GDSIIVia="true" Material="copper" Name="32"
    Thickness="0" Type="conductor"/>
    <Layer Color="#bd00bd" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="via" GDSIIVia="true" Material="copper" Name="33"
    Thickness="0" Type="conductor"/>
    <Layer Color="#3dffff" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="old" GDSIIVia="false" Material="copper" Name="34"
    Thickness="0" Type="conductor"/>
    <Layer Color="#3dffff" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="old" GDSIIVia="false" Material="copper" Name="35"
    Thickness="0" Type="conductor"/>
    <Layer Color="#3dffff" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="old" GDSIIVia="false" Material="copper" Name="36"
    Thickness="0" Type="conductor"/>
    <Layer Color="#3dffff" FillMaterial="FR4_epoxy" GDSDataType="0"
    TargetLayer="old" GDSIIVia="false" Material="copper" Name="55"
    Thickness="0" Type="conductor"/>
</Layers>
```

GDS Translation Using XML Control Files

Model creation from GDS files can be automated, or greatly accelerated, by using an XML control file. The XML control file allows for definition of materials, layer mapping, via processing, net tracing, and many other setup parameters, including the simulation setup. Using this mechanism, it is possible to translate into a ready-to-solve model. XML control files can also be used to establish templates for best practices across an organization, which can be useful to jump-start users less familiar with using HFSS for on-chip structures.

The control file schema is defined by multiple **.xsd* files, all of which can be found in the AnsysEM installation directory. An XML control file is referred to as an "instance document" for this schema, and can be validated using most text editors, or any number of 3rd-party XML editors. Additionally, virtually all programming languages provide libraries for validating and constructing XML files against schema definitions, making robust automation that much easier.

An XML control file is comprised of a root element, along with the following top-level elements:

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

- **Stackup** – used to define materials, layers, and vias.
- **Geometry** – can be used to add polygons to the translated output.
- **Boundaries** – for GDS translation, the schema applicable here are limited to point-based Circuit port creation and definition of HFSS extents.
- **Components** – selects the components' schema version.
- **CutoutSubdesign** – geometry clipping can be executed against a polygon using this element.
- **SimulationSetups** – defines HFSS simulation setups and sweeps.
- **ImportOptions** – specifies various import options.
- **GDS_CELL_RULES** – allows for control of which cells are imported.
- **GDS_NET_DEFINITIONS** – allows you to define VDD, Ground, and Signal nets.
- **GDS_COMPONENTS** – used to create component groups. This can be done automatically using the `GDS_AUTO_COMPONENT>` element.

From the following section, these elements are described in further detail. These sections do not cover all legal combinations of XML elements, but focus on common usage scenarios. Refer to an example of control file hierarchy for what these elements look like in practice.

Stackup

The `<STACKUP>` element is used to define materials, layers, and vias. These parameters can be duplicated:

- **<MATERIALS>** – may contain elements to define the permittivity, permeability, thermal properties, mechanical properties, conductivity, magnetic loss tangent, and dielectric loss tangent.
- **<ELAYERS>** – contains `LAYER` elements, which may be grouped under dielectrics, layers, or vias.

> **Note:**
>
> If no dielectrics or vias are present, you may use the `LAYERS` element without an overarching `ELAYERS` element.

For more information, refer to Via Grouping and Snapping in XML Control Files.

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

LayerAttr

An attribute group of <STACKUP> containing seven attributes (i.e., <NAME>, <TARGETLAYER>, <COLOR>, <MATERIAL>, <CONVERTPOLYGONTOCIRCLE>, <CONVERTPOLYGONTOCIRCLERATIO>, and <UNIONPRIMITIVES>).

ConvertPolygonToCircle

An attribute in the <LAYERATTR> attribute group.

Example

```
<Layers>
  <Layer Name="1" TargetLayer="Metal9" Type="conductor"
  Material="copper" Thickness="0.500000" Elevation="300"
  ConvertPolygonToCircle="true" ConvertPolygonToCircleRatio="0.5"/>
/Layers>
```

ViaLayer

A subcomponent group of <STACKUP> containing four elements (i.e., <DCTHICKNESS>, <CREATEVIAGROUPS>, <SNAPVIAGROUPS>, and <TSVPROPERTIES>) and four attributes (i.e., <GDSDATATYPE>, <STARTLAYER>, <STOPLAYER>, and<SOLVEINSIDE>).

TSVProperties

A subcomponent of <VIALAYER> containing two attributes (i.e., <THICKNESS> and <MATERIAL>).

Example

```
<Vias>
  <Layer Color="#2e43a0" Material="GDS_M1" Name="10"
  TargetLayer="TSV" SolveInside="false" StartLayer="UBMB"
  StopLayer="M1">
       <TSVProperties Thickness="5um" Material="EFFDIEL_1" />
  </Layer>
</Vias>
```

Geometry

The GEOMETRY element can be used to add polygons to the translated output. Most commonly, polygons impressed at this level are used to facilitate simulation setup details, such as the

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

definition of model extents or a mesh seeding region.

Example

```
<Geometry LengthUnit="um">
  <Polygon Layer="Outline">
    <Point x="8985.37" y="4967.452"/>
    <Point x="10636.639" y="4967.452"/>
    <Point x="10636.639" y="10370.592"/>
    <Point x="8985.37" y="10370.592"/>
    <Point x="8985.37" y="4967.452"/>
  </Polygon> </Geometry>
```

```
<Geometry LengthUnit="um">
  <Polygon Layer="user1" Name="seed region 1">
    <Point x="8985.37" y="4967.452"/>
    <Point x="10636.639" y="4967.452"/>
    <Point x="10636.639" y="10370.592"/>
    <Point x="8985.37" y="10370.592"/>
    <Point x="8985.37" y="4967.452"/>
  </Polygon>
</Geometry>
```

PolygonWithHoles

A subcomponent group of <GEOMETRY> containing three elements (i.e., <POINT>, <ARC>, and <HOLE>) and four attributes (i.e., <LAYER>, <NET>, <NAME>, and <PRECUTOUT>).

PreCutout

An attribute in the <POLYGONWITHHOLES> attribute group. Used to create outline and seeding region.

Example 1

```
<Geometry PreCutout = "false">
  <Polygon Layer="Metal9" PreCutout = "true">
    <Point x="133mm" y="-16mm"/>
    <Point x="148mm" y="-16mm"/>
    <Point x="148mm" y="0m"/>
    <Point x="133mm" y="0mm"/>
```


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

```
    <Point x="133mm" y="-16mm"/>
  </Polygon>
  <Polygon Layer="Metal9">
    <Point x="183mm" y="-16mm"/>
    <Point x="198mm" y="-16mm"/>
    <Point x="198mm" y="0m"/>
    <Point x="183mm" y="0mm"/>
    <Point x="183mm" y="-16mm"/>
  </Polygon>
</Geometry>
```

Example 2

```
<Geometry PreCutout = "false">
  <Polygon Layer="Metal9" PreCutout = "true">
    <Point x="133mm" y="-16mm"/>
    <Point x="148mm" y="-16mm"/>
    <Point x="148mm" y="0m"/>
    <Point x="133mm" y="0mm"/>
    <Point x="133mm" y="-16mm"/>
  </Polygon>
  <Polygon Layer="Metal9">
    <Point x="183mm" y="-16mm"/>
    <Point x="198mm" y="-16mm"/>
    <Point x="198mm" y="0m"/>
    <Point x="183mm" y="0mm"/>
    <Point x="183mm" y="-16mm"/>
  </Polygon>
</Geometry>
```

CriticalNet

A subcomponent of <GEOMETRY> containing one attribute (i.e., <NAME>).

CriticalNetGroup

A subcomponent group of <CRITICALNET> containing one element (i.e., <CRITICALNET>) and one attribute (i.e., <TOLERANCE>).

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

RemoveHoles

A subcomponent of <CRITICALNETGROUP> containing one element (i.e., <CRITICALNETGROUP>) and two attributes (i.e., <HOLEAREAMINIMUM> and <LENGTHUNIT>) used to remove small holes during GDS processing.

Example

```
<RemoveHoles HoleAreaMinimum="4.01" LengthUnit="mm">
  <CriticalNetGroup Tolerance="3">
    <CriticalNet Name="sig" />
  </CriticalNetGroup>
</RemoveHoles>
```

Boundaries

For GDS translation, the schema applicable in the <Boundaries> element are limited to point-based Circuit Port creation <CIRCUITPORTPT>and definition of HFSS Extents (<EXTENTS>).

CircuitPortPt

Ports in HFSS can take various forms, but for automation purposes, perhaps the most straightforward is defined by a pair of XY points and layers.

A list of these ports can be defined for creation in the Boundaries element of the XML.

Example

```
<Boundaries LengthUnit="mm">
  <CircuitPortPt Name="Port_1" x1="2.3" y1="3.4" Layer1="Top"
  x2="9.8" y2="8.7" Layer2="Bottom" />
  <CircuitPortPt Name="Port_2" x1="3.1" y1="4.5" Layer1="Top"
  x2="7.6" y2="6.5" Layer2="Middle" />
  <CircuitPortPt Name="Port_3" x1="5.6" y1="6.7" Layer1="Middle"
  x2="5.4" y2="4.3" Layer2="Bottom" />
</Boundaries>
```

Extents

HFSS Extents (i.e., the simulation region size) and airbox can be configured in the BOUNDARIES element *after* any <CIRCUITPORTPT> elements.

Example

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

```
<Boundaries LengthUnit="mm">
  <CircuitPortPt Name="Port_1" x1="2.3" y1="3.4" Layer1="Top"
  x2="8.2" y2="2.4" Layer2="Bottom" Z0="77"/>
    …
  <Extents Type="bbox" DielType="bbox" DielHorizFactor="0.01"
  AirboxHorizFactor="0.15" AirboxVertFactorPos="2.0"
  AirboxVertFactorNeg="2.0" UseRadiationBoundary="True"
  DielHonorPrimitives="false" AirboxTruncateAtGround="false" />
</Boundaries>
```

BoundaryConditions

A subcomponent of <BOUNDARIES> containing two elements (i.e., <CIRCUITPORTPT> and <EXTENTS>) and one attribute (i.e., <LENGTHUNIT>). Use to add boundary conditions.

Example

```
<Boundaries LengthUnit="mm">
  <CircuitPortPt Name="u2.67.dqd121" x1="9.48082" y1="10.003825"
  Layer1="UBM" x2="9.42582" y2="10.003825" Layer2="UBM" Z0="50"/>
  <CircuitPortPt Name="u1.8.dqd121" x1="9.48067" y1="6.180355"
  Layer1="UBM" x2="9.42567" y2="6.180355" Layer2="UBM" Z0="50"/>
  <Extents Type="bbox" DielType="bbox" DielHorizFactor="0.008"
  AirboxHorizFactor="0.16" AirboxVertFactorPos="1.55"
  AirboxVertFactorNeg="1.55" UseRadiationBoundary="True"/>
</Boundaries>
```

Components

A component group containing nine elements (i.e., <DIEPROPERTIES>, <SOLDERBALLPROPERTIES>, <PORTPROPERTIES>, <RLCMODEL>, <NPORTMODEL>, <NETLISTMODEL>, <SPICEMODEL>, <PINGROUP>, and <COMPONENTPORT>) and six attributes (i.e., <REFDES>, <PARTTYPE>, <PARTNAME>, <NUMPINS>, <NETS>, and <LENGTHUNIT>). The <COMPONENTS> element defines the component schema version.

Example

```
<Components schemaVersion="1.0"/>
```

COMPONENTPORT

A subcomponent group of <COMPONENTS> containing eight elements (i.e., <POSPIN>, <REFPIN>, <REFPINGROUP>, <REFNET>, <POSPINGROUP>, <REFPIN>,

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

<REFPINGROUP>, and <REFNET>) and two attributes (i.e., <NAME> and <Z0>). Use to port reference the closest pin on the net.

SPICEMODEL

A subcomponent group of <COMPONENTS> containing one element (i.e., <TERMINALPINMAPPING>) and five attributes (i.e., <MODELNAME>, <FILENAME>, <SUBCKT>, SOLVEINDEPENDENT>, and <REFERENCE NET>). <SPICEMODEL> is an element that contains the name and location of the selected spice (*.sp*) file. If the spice file's path is not given, it's assumed relative to the *.xml* file and that the files share a directory. The path can also be restricted to subdirectories of the *.xml* file's path. If the spice file's path is relative when an Ansys Electronic Desktop (*.aedt*) file is created, the spice file will be relocated to the *.aedt* file's directory, so that the path remains relative. This is not the case if an absolute path is set.

Example - Absolute Path

```
<GDS_COMPONENT>
  <GDS_PIN Name="3" x="-19.925" y="11.385" Layer="Top"/>
  <GDS_PIN Name="4" x="6.915" y="9.35" Layer="Top"/>
    <Component RefDes="C1" PartName="12344-CAP" PartType="Capacitor">
      <SpiceModel ModelName="2pin_Ind‟ FileName="F:/IC_ON_
      PKG/testcase/hbm/testcomponentmodels2port_ind.sp" SubCkt="Test_
      2pin_Ind">
                <TerminalPinMapping term="1" pin="3"/>
                <TerminalPinMapping term="2" pin="4"/>
      </SpiceModel>
    </Component>
</GDS_COMPONENT>
```

Example - Relative Path

```
<GDS_COMPONENT>
  <GDS_PIN Layer="Metal9" Name="3" x="-0.8" y="0"/>
  <GDS_PIN Layer="Metal9" Name="4" x="0.8" y="0"/>
    <Component PartName="12344-CAP" PartType="Capacitor" RefDes="C1">
      <SpiceModel FileName="datafiles/2port_ind.sp" ModelName="2pin_
      Ind" ReferenceNet="4" SolveIndependent="false" SubCkt="Test_
      2pin_Ind">
                <TerminalPinMapping pin="3" term="1"/>
                <TerminalPinMapping pin="4" term="2"/>
      </SpiceModel>
    </Component>
```


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

```
</GDS_COMPONENT>
```

CutoutSubdesign

Geometry clipping can be executed against a polygon using the <CUTOUTSUBDESIGN> element.

Example

```
<CutoutSubdesign>
  <Polygon>
    <Point x="8985.37um" y="4967.452um"/>
    <Point x="10636.639um" y="4967.452um"/>
    <Point x="10636.639um" y="10264.592um"/>
    <Point x="8985.37um" y="10264.592um"/>
    <Point x="8985.37um" y="4967.452um"/>
   </Polygon>
</CutoutSubdesign>
```

SimulationSetups

The <SIMULATIONSETUPS> element contains an <HFSSSETUP> element for each setup being defined. Refer to this example of Control File Schema (from Simulation Setup) for an example of the schema in action.

HFSSSetup

A subcomponent group of <SIMULATIONSETUPS> containing two elements (i.e., <HFSSSIMULATIONSETTINGS> and <HFSSSWEEPDATALIST>) and two attributes (i.e., <NAME> and <SCHEMAVERSION>).

HFSSSimulationSettings

An element group of <HFSSSETUP> containing fifteen elements (i.e., <ENABLED>, <SAVEFIELDS>, <SAVERADFIELDSONLY>, <HFSSADAPTIVESETTINGS>, <HFSSINITIALMESHSETTINGS>, <HFSSDEFEATURESETTINGS>, <HFSSVIASETTINGS>, <HFSSCURVEAPPROXSETTINGS>, <HFSSADVANCEDMESHSETTINGS>, <HFSSMODELLINGSETTINGS>, <HFSSSETUPOUTPUTVARIABLES>, <HFSSPORTSETTINGS>, <HFSSSOLVERSETTINGS>, <HFSSDCRSETTINGS>, and <HFSSMESHOPERATIONS>).

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

HFSSSweepDataList

An element group of <HFSSSIMULATIONSETTINGS> containing one element (i.e., <FREQSWEEP).

Example

```
<HFSSSweepDataList>
  <FreqSweep>
    <Name>Sweep 1</Name>
    <Enabled>True</Enabled>
    <UseQ3DForDC>False</UseQ3DForDC>
    <Interpolating>
      <LinearStep>
      <Start>500mhz</Start>
      <Stop>10ghz</Stop>
      <Step>500mhz</Step>
      </LinearStep>
    </Interpolating>
  </FreqSweep>
</HFSSSweepDataList>
```

RemoveIslandsTolerance

An optional attribute of <ELayers> that exposes the **Remove small islands** command in the XML control file for GDS translation.

Example

```
<ELayers LengthUnit="mm" RemoveIslandsTolerance="1mm2">
  <Layers>
    <Layer Name="metal1" Material="metal1" Thickness="0.002"
Type="conductor"/>
  </Layers>
</ELayers>
```

ImportOptions

The <IMPORTOPTIONS> element specifies various import options in one line.

Example

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

ImportOptions

The <IMPORTOPTIONS>> element specifies various import options in one line.

Example

```
<ImportOptions Flatten="true" GDSIIConvertPolygonToCircles="false"
ImportDummyNet="false"/>
```

> **Note:**
>
> <ImportDummyNet="false"> deletes all metal that has not been assigned a net. This may result in all the metal in the design being deleted if conductive layers are set to assign nets but none are assigned.

GDS_CELL_RULES

The <GDS_CELL_RULES> element allows for control of which cells are imported. Substitutions are supported using any number of Wildcard characters (*) and any number of single characters (?). Cell names are case-insensitive and are separated by white space (including tabs and end-of-lines).

Cells are divided into three elements:

- **<BLACK_BOX_CELLS>** – no geometry is imported on these cells. They are deleted prior to net assignment and flattening.
- **<GRAY_BOX_CELLS>** – geometry on these cells are only assigned nets if they are connected to other cells. This assignment could be generated by either <WHITE_BOX_CELLS> or <GDS_NET_DEFINITIONS>. In either case, all geometries not assigned a net are deleted. If neither are defined, it is assumed that all cells except <BLACK_BOX_CELLS> and <GRAY_BOX_CELLS> can be used to generate net assignments.
- **<WHITE_BOX_CELLS>** – geometry on these cells is used to generate net assignments, unless <GDS_NET_DEFINITIONS> rules are defined. Metal in other cells may be assigned nets if they connect to the geometry in <WHITE_BOX_CELLS>.

Example

```
<GDS_CELL_RULES>
  <BLACK_BOX_CELLS> Bl* Bad </BLACK_BOX_CELLS>
  <GRAY_BOX_CELLS> Gra? </GRAY_BOX_CELLS>
  <WHITE_BOX_CELLS>White</WHITE_BOX_CELLS>
</GDS_CELL_RULES>
```


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

GDS_NET_DEFINITIONS

Nets are defined as one of:

- **<VDD_NETS>**
- **<GND_NETS>**
- **<SIGNAL_NETS>**

All three allow you to select which nets/text you want to import. They also allow the assignment of net names based on a location (layer name, x microns, y microns) . Additional elements under `<GDS_NET_DEFINITIONS>` are:

- **<USE_TEXT_FROM_HIERARCHY_LEVEL>** – the text at specified hierarchy levels is used to generate nets. 0 is the top. White space denotes a separation. A dash (-) indicates all intermediate layers. "MAX" indicates all hierarchical elements under the previous entry and should always follow a dash.
- **<USE_TEXT_FROM_CELLS>** – the text in specified cells is used to generate nets. Wild cards are allowed.
- **<USE_TEXT_FROM_HIERARCHY_BLOCK>** – the text of an exactly placed hierarchy block is used to generate nets. Wild cards are not allowed.

Rules defined in all three of these elements may be applied at once.

Example

```
<GDS_NET_DEFINITIONS NET_NAME_CASE_SENSITIVE="false" USE_TOP_LEVEL_
TEXT_ONLY="false">
  <USE_TEXT_FROM_HIERARCHY_LEVEL>0 2 4-MAX</USE_TEXT_FROM_HIERARCHY_
  LEVEL>
  <USE_TEXT_FROM_CELLS>TextC*</USE_TEXT_FROM_CELLS>
  <USE_TEXT_FROM_HIERARCHY_BLOCK> Top/Text </USE_TEXT_FROM_HIERARCHY_
  BLOCK>
  <VDD_NETS>MODE {MODE? MODE@ layer_7 5000.0 7000.0} VBA* VCC@layer_1
  4000.0 3000.0</VDD_NETS>
  <GND_NETS>RFO</GND_NETS>
  <SIGNAL_NETS>SENSdin</SIGNAL_NETS>
</GDS_NET_DEFINITIONS>
```


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

Note:

Though all options are displayed, typically they are not used together, and they can be contradictory. Net creations for all <GDS_NET_DEFINITIONS> stem from text defined in the GDSII file or from point locations included in <VDD_NETS>, <GND_NETS>, and <SIGNAL_NETS>. If the rules are self-contradictory or too restrictive it is possible that no nets are generated.

Auto Net Tracing in GDS/OASIS Designs

If designs in **GDS/OASIS format** have no text labels defined, tracing is impossible. Net tracing requires text labels and custom rules in the XML control file. Auto net tracing allows tracing of the largest nets (i.e., the largest physically connected elements with the greatest total area.

Configuration

To enable auto nets, use the following attributes in the <GDS_NET_DEFINITIONS> tag of the XML control file:

- **AUTO_NETS="N"** – N is a positive integer specifying how many largest nets to trace.
- **AUTO_NETS_PREFIX="name"** – Optional. Defines the prefix for auto net names. Default prefix is AutoNet_*.

Auto nets are listed in descending order of size.

Example

```
<GDS_NET_DEFINITIONS AUTO_NETS="10" AUTO_NETS_PREFIX="MYNE
<GDS_NET_DEFINITIONS AUTO_NETS="5"/>
```

The system will first trace nets defined in **GND_NETS**, then attempt to trace the N largest nets as specified.

Example 2

The following snippet shows example output entries from <GDS_NETS> after auto net tracing:

```
<GDS_NETS>
PADS_PLL0_PVDD1P5@ubmb 2150.87 ...
...
</GDS_NETS>
```

These entries are not required for configuration, but they can be useful for verification.

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

GDS_COMPONENTS

A component group containing two elements (i.e., <GDS_COMPONENT> and <GDS_AUTO_COMPONENT>) and one attribute (i.e., <LENGTHUNIT>). Using <GDS_COMPONENT>, components can be created using XML. Component Groups can also be created manually, or automatically using the <GDS_AUTO_COMPONENT> element. When using <GDS_AUTO_COMPONENT>, pins are created from all geometry on a given layer. Those pins are then partitioned into component groups, based on the provided tolerance. Component type, solderball definitions, and reference size can also be defined. Finally, ports can be created on pins belonging to specified nets.

Example 1

```
<GDS_COMPONENTS LengthUnit="um">
  <GDS_AUTO_COMPONENT Layer="UBM" Tolerance="100">
    <DieProperties Type="Flip chip" Orientation="Chip down"
    Height="0.0"/>
    <SolderballProperties Shape="Cylinder" Diameter="6um"
    Height="10um" Material="solder"/>
    <PortProperties ReferenceOffset="0" ReferenceSizeAuto="True"/>
    <PadPort Net="HBM_DQd*"/>
  </GDS_AUTO_COMPONENT>
</GDS_COMPONENTS>
```

Example 2 (Pad Ports)

```
<GDS_COMPONENT>
  <GDS_PIN Name="0" x="9425.670000" y="6276.355000" Layer="UBM"/>
  <GDS_PIN Name="1" x="9480.670000" y="6276.355000" Layer="UBM"/>
  <GDS_PIN Name="2" x="9535.670000" y="6276.355000" Layer="UBM"/>
  <GDS_PIN Name="3" x="9590.670000" y="6276.355000" Layer="UBM"/>
    …
<Component RefDes="U1" PartName="DIE1" PartType="IO">
<DieProperties Type="Flip chip" Orientation="Chip down" Height="0"/>
<SolderballProperties Shape="Cylinder" Diameter="6um" Height="10um"
Material="solder"/>
<PortProperties ReferenceOffset="0" ReferenceSizeAuto="True"/>
<Port Name="u1.4.dqd120">
  <Pin Name="4"/>
  <Pin Name=""/>
</Port>
```


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

```
  <Port Name="u1.5.dqd112">
    <Pin Name="5"/>
    <Pin Name=""/>
</Port>
```

Example 3 (Netlist)

A model can be added to a component using XML.

```
<GDS_COMPONENTS LengthUnit="um">
  <GDS_COMPONENT>
    <GDS_PIN Name="1" x="-19.925" y="11.385" Layer="Top"/>
    <GDS_PIN Name="2" x="6.915" y="9.35" Layer="Top"/>
    <Component RefDes="C1" PartName="12344-CAP" PartType="Capacitor"
    >
      <NetlistModel>
                <Netlist>C4 1 2 1e-12</Netlist>
      </NetlistModel>
    </Component>
  </GDS_COMPONENT>
</GDS_COMPONENTS>
```

Example 4 (Spice)

```
<GDS_COMPONENT>
  <GDS_PIN Name="3" x="-19.925" y="11.385" Layer="Top"/>
  <GDS_PIN Name="4" x="6.915" y="9.35" Layer="Top"/>
  <Component RefDes="C1" PartName="12344-CAP" PartType="Capacitor">
    <SpiceModel ModelName="2pin_Ind‟ FileName="F:/IC_ON_
    PKG/testcase/hbm/testcomponentmodels2port_ind.sp" SubCkt="Test_
    2pin_Ind">
      <TerminalPinMapping term="1" pin="3"/>
      <TerminalPinMapping term="2" pin="4"/>
    </SpiceModel>
  </Component>
</GDS_COMPONENT>
```

GDS_TEXT_PINS

A subcomponent option of <GDS_COMPONENT> that allows automatic creation of pins from text labels in the design. This option uses the names and coordinates of text labels to define pins,

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

removing the need to manually list pin locations.

Under <GDS_TEXT_PINS>:

- Add text names individually.
- Use text wildcards to group multiple labels, similar to <GDS_NETS>.
- Match texts on the design with the provided names/wildcards.
- Use text coordinates as pin locations.

Pin creation logic remains the same as with <GDS_PIN>, including handling of overlapping pins. Duplicate pin names are automatically incremented (e.g., N7_GND, N7_GND_1, N7_GND_2).

Examples

```
<GDS_COMPONENT>
<GDS_TEXT_PINS Text="N7*" Layer="M17"/>
<Component RefDes="U1"/>
</GDS_COMPONENT>

<GDS_COMPONENT>
<GDS_TEXT_PINS Text="HBM*" Layer="M17"/>
<Component RefDes="U2"/>
</GDS_COMPONENT>

<GDS_COMPONENT>
<GDS_TEXT_PINS Text="*HBM*" Layer="M6"/>
<Component RefDes="U5" PartType="IO" PartName="BGA"/>
</GDS_COMPONENT>

<GDS_COMPONENT>
<GDS_TEXT_PINS Text="*SoC*" Layer="M6"/>
<Component RefDes="U6" PartType="IO" PartName="BGA"/>
</GDS_COMPONENT>
```

Both <GDS_TEXT_PINS> and <GDS_PIN> can coexist under the same component.

```
<GDS_COMPONENT>
<GDS_TEXT_PINS Text="TEXT_2" Layer="M17"/>
<GDS_TEXT_PINS Text="PIN_*" Layer="M17"/>
<GDS_PIN Name="PIN_1" x="1um" y="1um" Layer="M17"/>
<GDS_PIN Name="PIN_2" x="1um" y="2um" Layer="M17"/>
<Component RefDes="U1"/>
</GDS_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:

Control file changes are backwards compatible. If text names are not unique, they are incremented automatically (e.g., `pin`, `pin_1`, `pin_2`).

GDS_AUTO_COMPONENT

A subcomponent group of `<GDS_COMPONENTS>` containing four elements (i.e., `<DIEPROPERTIES>`, `<SOLDIERBALLPROPERTIES>`, `<PORTPROPERTIES>`, `<AUTOCOMPONENTPORT>`) and two attributes (i.e., `<LAYER>` and `<TOLERANCE>`).

AUTOCOMPONENTPORT

A subcomponent group of `<GDS_AUTO_COMPONENT>` containing two elements: `<GDS_COMPONENT>` and `<GDS_AUTO_COMPONENT>`.

Use to port reference the closest pin on the net.

Example - Auto

```
<AutoComponentPort>
  <PosNet Name="HBM_DQd*"/>
  <RefNet Name="VSS"/>
</AutoComponentPort>
```

Example - Manual

```
<Component RefDes="U2" PartName="DIE2" PartType="IO">
  <DieProperties Type="Flip chip" Orientation="Chip down"
  Height="0"/>
  <SolderballProperties Shape="Cylinder" Diameter="6um" Height="10um"
  Material="solder"/>
  <PortProperties ReferenceOffset="0" ReferenceSizeAuto="True"/>
  <ComponentPort Name="u2.38.dqd119">
    <PosPin Name="38"/>
    <RefNet Name="VSS"/>
  </ComponentPort>
</Component>
```


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

ODB++ Translator

ODB++ is an ASCII open-format developed by Mentor Graphics to capture CAD/EDA, assembly, and PCB fabrication information. It has become the proprietary industry standard for formatting and transferring PC board data.

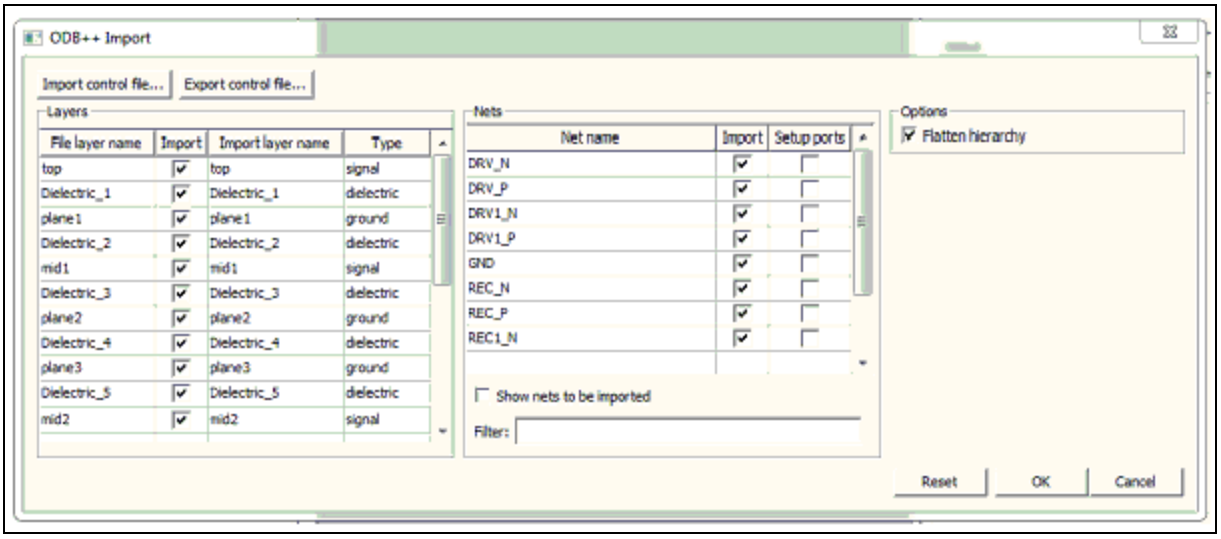
AnsTranslator, allows you to utilize ODB++ and ANF file formats to perform analyses of printed circuit boards in support of Ansys SIwave™. **AnsTranslator** takes as input an ODB++ v7.1 job structure and produces as output an ANFv4 file or an ANFv2 file and an Ansys SIwave™ component file (.cmp).

- The input to **AnsTranslator** is an ODB++ v7.1 job structure (either as a compressed .tgz file or an uncompressed top folder) and an optional control file (.xml).
- **AnsTranslator** can export to an edb directory or to a control file.
- The output from **AnsTranslator** is either an EDB directory or ANFV2 file and a component file (.cmp). ANFV4 is also supported. A log file (.log) is also produced.
- The information translated is a collection of geometric data that is sufficient to perform a useful analysis of the represented circuit for Ansys SIwave™ designs (which also supports Ansys HFSS™, Ansys Q3D Extractor®, and TPA). Note the **AnsTranslator** process is subject to whatever limitations exist for the ANF format to represent ODB++ content.
- If the board outline is included in the profile file, it are translated onto an assembly layer called "outline". This same polygon are used to translate negative-plane data to a positive-plane shape. If the board outline is composed entirely of disjoint segments, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform attempts to stitch the data together so long as the end points are within a nanometer of each other.
- The data can be modified through the Control File.

Importing ODB++ in Ansys Electronics Desktop

The ODB++ translator is a link to an external DLL that can be accessed by clicking **File** > **Import** > **ODB++** which opens the import window.

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



Import and Export Control Buttons

- **Import control file** opens a window which allows a user to import a selected control file.
- **Export control file** writes an xml control file based on the information in the window to the location specified.

Layers Pane

- **Import** check box specifies which layers to import.
- **Import layer** name can be edited.
- **Type** can be changed and identifies the layer type that are used to place each layer in the file (not editable if the layer already exists; for layers that don't exist, click in the column to see a drop-down menu of choices).

Nets Pane

- **Import** check box specifies which nets to import.
- The **Nets** grid control allows users to select nets to import, and indicate if pins on that net should be imported as ports.
- Pins that belong to nets with a check in the **SetupPorts** column become ports when imported.
- **Show nets to be imported** moves nets to be imported to the top of the grid control.
- **Filter** involves enhanced wild card filtering is used to select which nets are shown in the window by using wild cards and characters; the two wild cards supported are * and ? (See **Regular Expressions for Nets** below).

Options Pane

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

- **Flatten hierarchy** allows users to select importing the ODB++ with components or flattening the components.

When you have completed selections, click **OK** and the file is imported into the active Layout window.

Regular Expressions for Nets

You can use regular expressions to search for nets. These match a text pattern and use special constructions to represent strings or single characters.

To use a regular expression, enter the name of the object you want to select in the field, using wild cards when selected. The following regular expressions are supported:

*****	Matches any sequence of characters. For example, **net*** selects objects **net_1a**, **net_1b**, **netLength**.
?	Matches any character. For example, **net_?** selects objects **net_1**, **net_2**, **net_a**, **net_b**.

These match a text pattern and use special constructions to represent strings or single characters (e.g., net* selects all pins in nets that start with “net”, net*:1* selects pins with name that start with “1” in the nets that start with name “net”, A*:3? selects pins 31, 32, 33, 34 etc in nets, A1, ACC, AGND, AGND2).

Control File Part Mapping

During translation, it is often appropriate to augment available data for the purposes of simulation. One area that is often vital is component classification and value assignment; this supplementary information can be added using an XML Control File with a Components/TypeMap and/or ODBPropertyMap.

- The *TypeMap* element provides a sequence of either *Part* or *RefDes* elements.
- Each element specifies a Perl Regular Expression.
- A component's type and value are matched sequentially using the *TypeMap*, with *Part* elements evaluating against a part name and *RefDes* elements against the reference designator.

For example, a Control File mapping the type of components with reference designators beginning in 'L' to an inductor, 'R' to a resistor, and 'C' to a capacitor, is implemented as:

```
<?xml version="1.0" encoding="UTF-8"?>
<c:Control schemaVersion="1.0 "
xmlns:c="http://www.ansys.com/control">
  <Components schemaVersion="1.0 ">
```


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

```
    <TypeMap>
      <RefDes Type="Inductor " RegEx="L.*"/>
      <RefDes Type="Resistor " RegEx="R.*"/>
      <RefDes Type="Capacitor " RegEx="C.*"/>
    </TypeMap>
  </Components>
</c:Control>
```

The regular expressions in this example are mutually exclusive and, thus, do not rely on order for evaluation. A more complicated *TypeMap*, using part names, reference designators, and relying on order is:

```
<TypeMap>
  <RefDes RegEx="L.*" Type="Inductor"/>
  <RefDes RegEx="R.*" Type="Resistor" SR_Value="25ohm" />
  <Part RegEx="100nF " Type="Capacitor" SC_Value="100nF" SR_
  Value="0.1ohm" />
  <Part RegEx=".*nF " Type="Capacitor" SC_Value="0.1uF" SR_
  Value="0.1ohm" SL_Value="0.1nH" />
  <RefDes RegEx="C.*" Type="Capacitor" SC_Value="1uF"/>
</TypeMap>
```

The following mappings should result:

Part Name	Refdes	Mapped To
AA-0.1nF	C21	Capacitor: (0.1uF,0.1ohm,0.1nH)
AB-1000nF-T	C22	Capacitor:(1uF)
100nF	C23	Capacitor:(100nF,0.1ohm)
100nF	L-C24	Inductor

Information regarding component type and values can be specified in the control file. If the component information is not in the control file, the translator looks for optional properties in the ODB++ component files if the component has 2 pins. If the component records in these files have property records TYPE and VALUE, the contents of these properties are used to determine component information. All comparisons are case independent.

The following table shows the type assigned to the component based on the TYPE property content:

TYPE property	Type assigned to the component
"resistor" or "embedded resistor"	resistor

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

TYPE property	Type assigned to the component
"inductor"	inductor
"capacitor" or "embedded capacitor"	capacitor
any other TYPE content	**the content of VALUE is used to deduce the type**
1. content does not begin with a number	component is assumed to be discrete
2. content begins with a number	
• without any following text	resistor
• last letter is "h"	inductor
• last letter is "f"	capacitor
• following text includes "ohm"	resistor
• last letter is a unit abbreviation ("p", "n", "u", "m", "k", "meg", "g", or "t")	resistor

ODBPropertyMap

If an ODBPropertyMap is not defined in the .xml file types and values of component are assigned primarily through the TypeMap. Any unmatched components then has their type and value set on the component PRP with key *Type* in the component_top and component_bottom file and the values on the PRP key called *Value* or *Val*. If additional control is selected one can define the ODBProperty map.

If one wishes to use the TypeMap as default values one can now set the "PrioritizeOverTypeMap" attribute to true in the ODBPropertyMap. From the following case, if the a component R1 had a resistance value set in the VALUE key of "1K": with the flag set to true the resistance is set to 1000 ohm, if the flag is not set the resistors value are 25ohm.

Fields are listed in priority order and the names are compared without case sensitivity. That is the field named Type takes precedence over all the other fields and it matches "TYPE", "type", etc. Because the TypeOnly field is true it are only used to set the type and should not correspond to a value. If this is being used for R1, it is likely "resistor". If a field is not "TypeOnly" it are used to infer an RLC type. It is important to make your "TypeOnly" first in the list if you need to guarantee it are used. At present only 2 pin components with properties are supported.

The SeparationCharacter attribute takes a string of characters to cut at. For example, if the value supplied are in the form "220K/5%" and "3.9K,1%", set SeparationCharacter = ", /".

This string cuts at commas, spaces and the back slash.

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<c:Control xmlns:c="http://www.ansys.com/control"
schemaVersion="1.0">
  <Components schemaVersion="1.0">
    <TypeMap>
```


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

```
      <RefDes RegEx="L.*" Type="Inductor"/>
      <RefDes RegEx="R.*" Type="Resistor" SR_Value="25ohm" />
      <Part RegEx="100nF" Type="Capacitor" SC_Value="100nF" SR_
      Value="0.1ohm" />
      <Part RegEx=".*nF" Type="Capacitor" SC_Value="0.1uF" SR_
      Value="0.1ohm" SL_Value="0.1nH" />
      <RefDes RegEx="C.*" Type="Capacitor" SC_Value="1uF"/>
    </TypeMap>
    <ODBPropertyMap PrioritizeOverTypeMap ="true">
      <ODBProperty Name="Type" TypeOnly="true"/>
      <ODBProperty Name="Value" SeparationCharacters = ", /"/>
      <ODBProperty Name="Val"/>
      <ODBProperty Name="InstPar" SeparationCharacters = ", /"/>
    </ODBPropertyMap>
  </Components>
</c:Control>
```

Exporting Layout Design Data

Design and analysis data from the Ansys Electronics Desktop (AEDT)™ electronics systems design platform can be exported to other systems in a variety of industry-standard data formats. Note the default export file name is made up of the project name followed by a space followed by the circuit/subcircuit name.

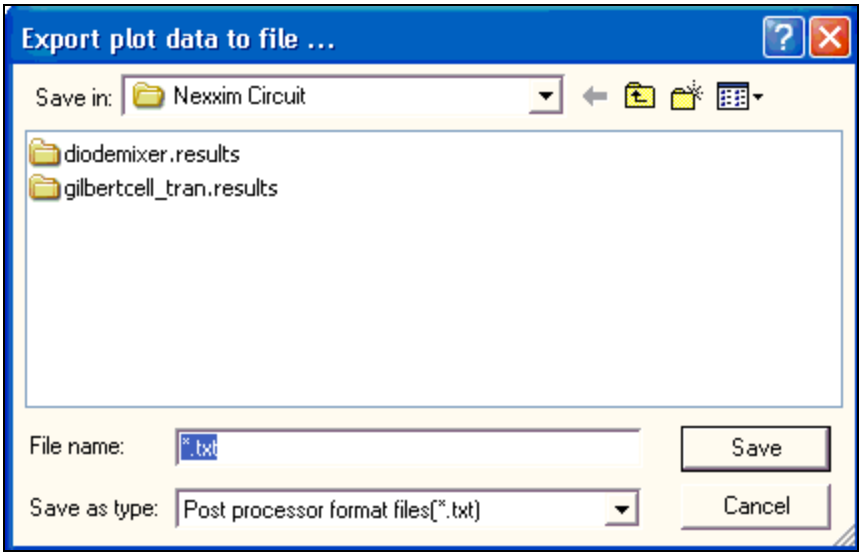
Exporting Data Table Files

Follow these steps to export simulation results in ASCII tabular format.

1. Open the plot you want to export.

2. From the **Report2D** menu, select **Export to File**. (Alternatively, right-click the data table, then click **Export to File**.) to open the **Export plot data to file** window.

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

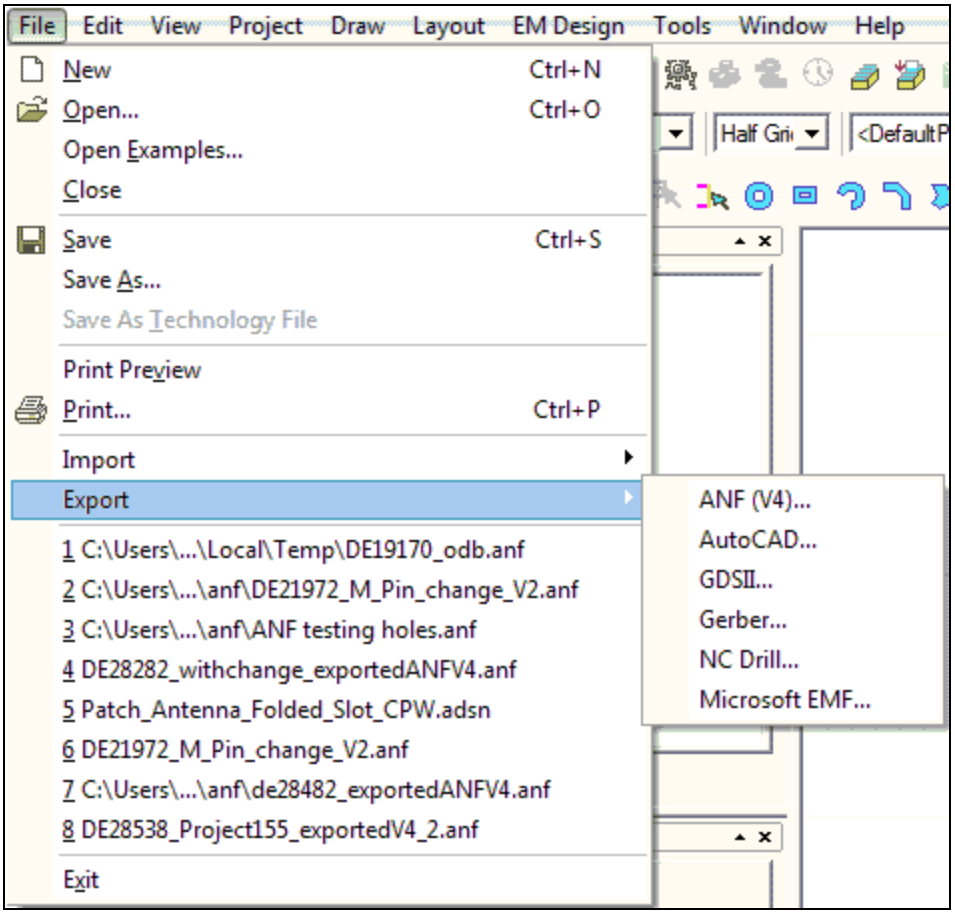


3. Browse to the directory, then give the file a name. Click **Save**.

Exporting Layout Data

Using the **File** > **Export** command, you can export layout data using the following file formats/types:

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



- On physical layers such as silkscreen and signal layers, property displays are drawn with Hershey fonts and possess a constant size. These property displays are exported from the Ansys Electronics Desktop (AEDT)™ electronics systems design platform to GDSII, Gerber, and DXF formats.
- On non-physical layers such as symbol layers, property displays do not possess a constant size — they are scaled during zooming. As a result, property displays on non-physical layers do not export from AEDT.

To export layout data:

1. From the **File** menu, click **Export**. Then select the type of file to export to open an explorer window.
2. Select a file format and click **Save**.

If you select Ansys Neutral or Microsoft Enhanced the file is saved. If you select a different format, one of the following window opens:

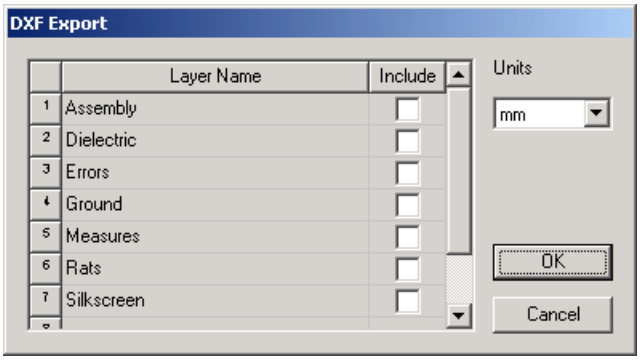
- DXF Export
- GDSII Export

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

- Gerber Export
- NC/Drill Export

DXF Export

To export layout data, on the **File** menu, select **Export** > **AutoCAD** to open an explorer window. Selecting **AutoCad DXF File** opens the **DXF Export** window**.**



- Choose a unit of measurement on the **Units** drop-down menu (cm, ft, in, km, meter, mil, nm, uin, um). Default type is *mm*.
- To save a **Layer Name**, check its corresponding **Include** area, and click **OK.**

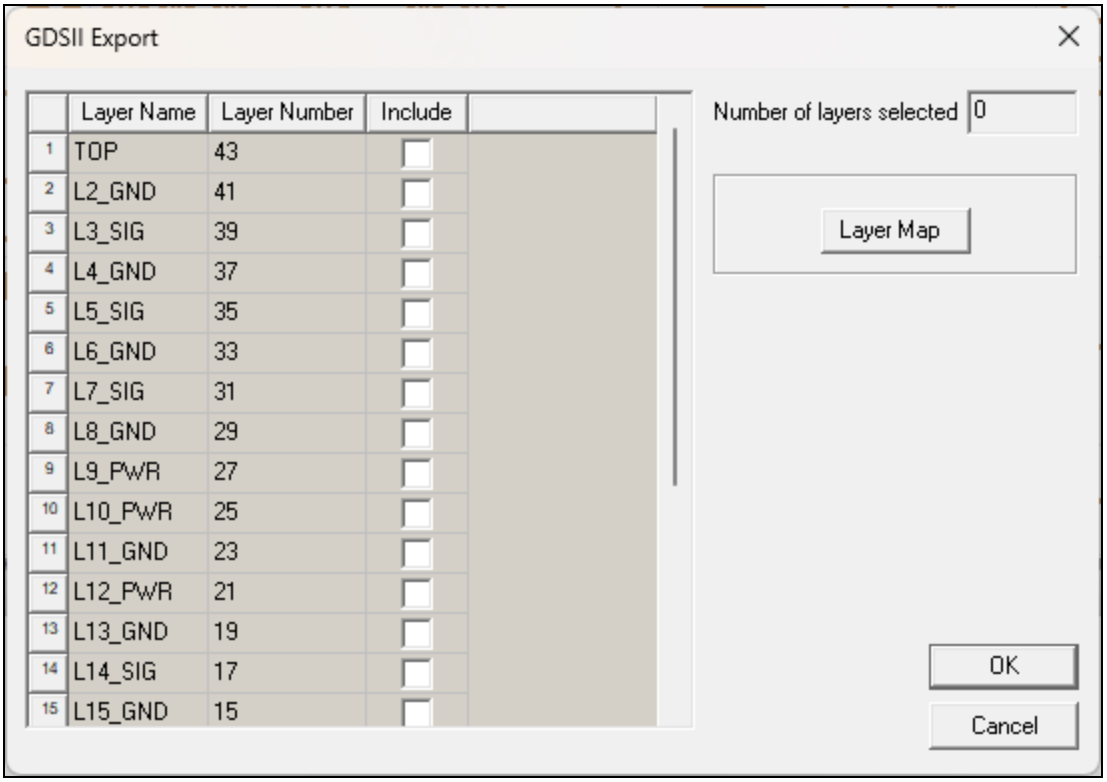
During export, a polygon with voids is converted into a single polygon with no holes where bridges connect the voids to the outer contour.

Text that is Hershey font are exported as polygons, that is, the text is thickened.

GDSII Export

To export layout data, on the **File** menu, select **Export** > **GDSII** to open an explorer window. Then select **GDSII** or **GDSII (Stream)** to open the **GDSII 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.

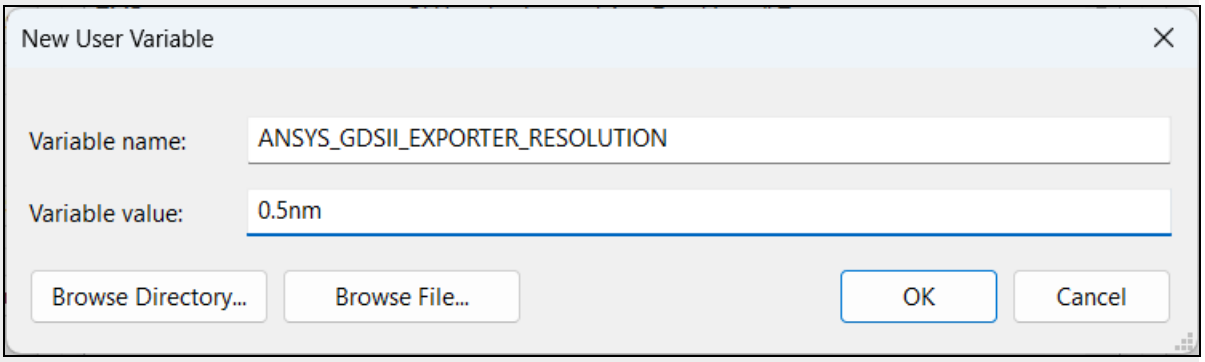


Text that is Hershey font are exported as polygons, that is, the text is thickened.

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 Ansys Electronics Desktop (AEDT)™ electronics systems design platform assumes exported GDSII files require a resolution of 1nm. If users export GDSII files with a resolution less than 1nm, the user can change the value of the parameter by creating the environment variable `ANSYS_GDSII_EXPORTER_RESOLUTION` and entering an appropriate value (e.g., `ANSYS_GDSII_EXPORTER_RESOLUTION=0.5nm`).



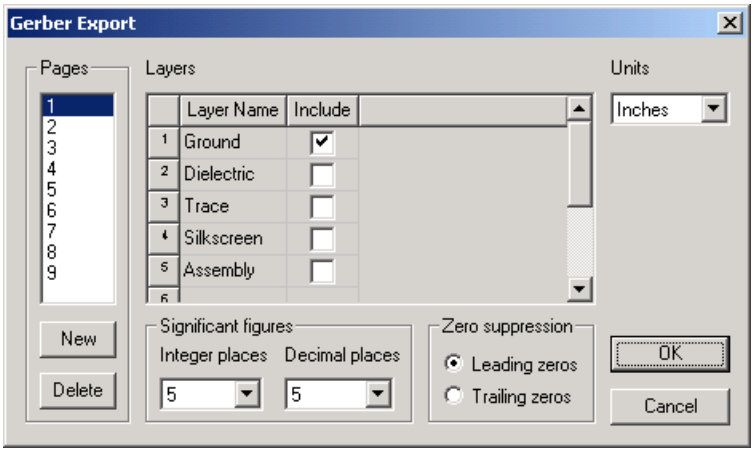
Gerber Export

When you export to Gerber from a layout in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform, each layer is treated as a separate “page” and each page is exported to a separate file. Each file is named by appending the layer/page number to the export file name that is selected by the user.

- You can export more than one layer/page to a Gerber file by selecting additional layers in the Gerber Export window. Header information in each exported file lists the layers that are included in the file.
- You can suppress the exporting of a particular Gerber layer by deleting the page(s) in the Gerber Export window. Only those pages that are displayed in the Gerber Export window are exported.

To export layout data, on the **File** menu, select **Export** > **Gerber** to open an explorer window. Name and save the .Gerber file. Then, select **Gerber** to open the **Gerber 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.

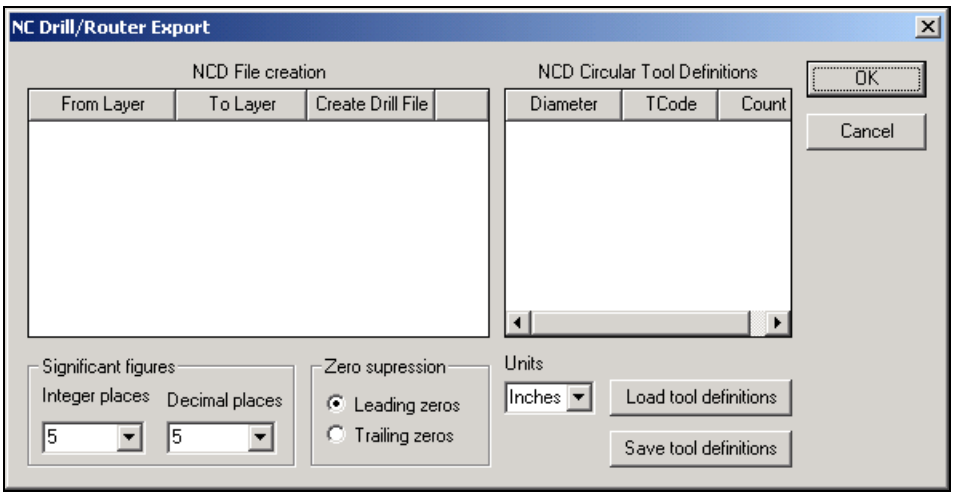


- Click **New** to add pages or Click **Delete** to remove pages.
- Check an **Include** area for each Layer you wish to save.
- Set **Units** to Inches or Millimeters.
- Select the appropriate **Zero suppression** setting.
- Select the appropriate number of **Significant figures** for **Integer** and **Decimal** places.
- Click **OK**.

Text that is Hershey font are exported as polygons. That is, the text is thickened. AEDT exports Gerber RS274-X format.

NC Drill/Router Export

To export layout data, on the **File** menu, select **Export** > **NC Drill** to open an explorer window. Selecting **NC Drill File** opens the **NC Drill/Router 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.

- Click **Load tool definitions** to load and view definitions.
- Click **Save tool definitions** to save definitions.
- Select the appropriate **Zero suppression** setting.
- Select the appropriate number of **Significant figures** for **Integer** and **Decimal** places, and click **OK**.

Exporting Graphics Files on the Layout 3D Editor

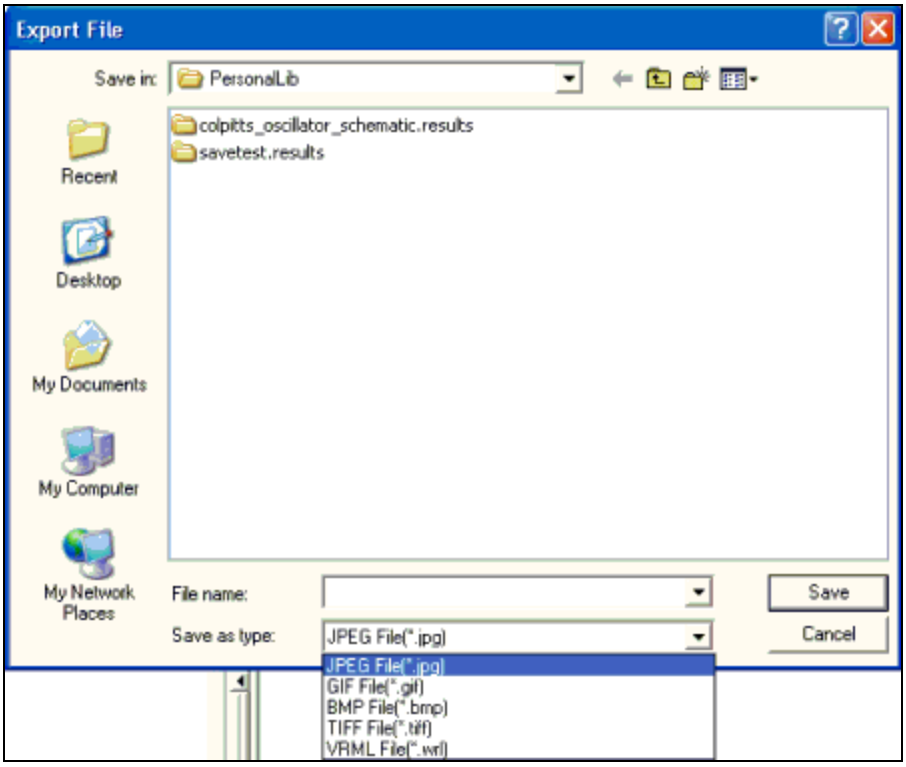
You can export the following graphics formats on the 3D layout editor:

Extension	Contents
.bmp	Bitmap files.
.gif	Graphics Interchange Format files.
.jpeg	Joint Photographics Experts Group files.
.tiff	Tagged Image File Format files.
.wrl	Virtual Reality Modeling Language (VRML) files.

To export a file to a graphics format:

1. From the **Edit 3D** menu, select **Export** to save the file in a graphics format. The Export File 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.



2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File Name** area.
4. Select the appropriate graphics file format on the **Save as type** pull-down list.
5. Click **Save**. The file is exported to the specified location as a graphics file.

Exporting ANF Layout or Schematic Data

You can export design data in the Ansys neutral file format (.anf) from either the layout or the schematic editors in the the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. From the **File** menu, select **Export** > **ANF (V4)** to open an explorer window. Name and save the .anf file.

For a reference to the Ansys Neutral File format specification, see Ansys Neutral File Format.

Importing Circuit Models

Ansys provides support for imported models and libraries as components in the **Schematic Editor**. You can import the following models:

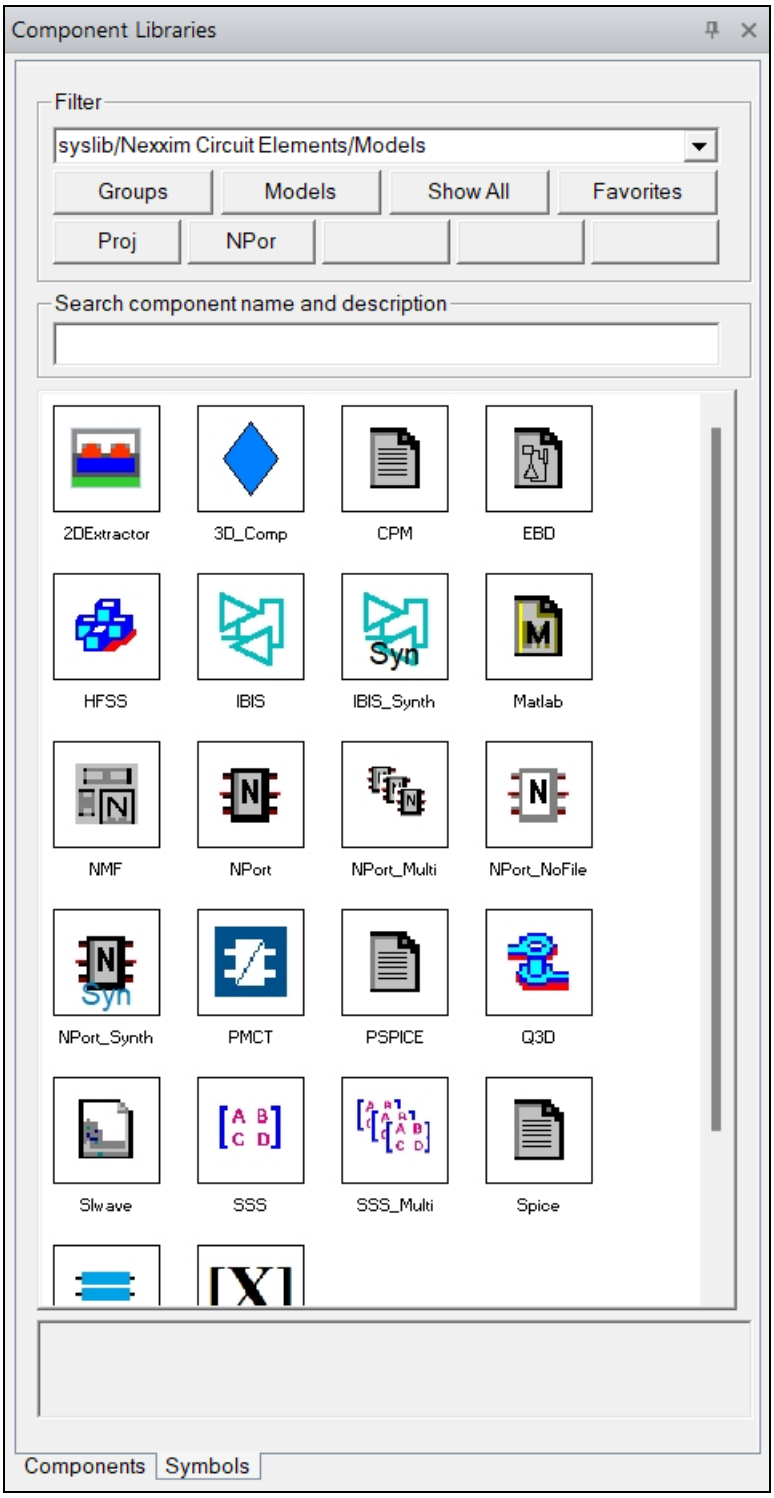
- SPICE components and libraries
- IBIS buffer elements
- W-elements

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

- State-space models
- X-parameter elements
- Verilog models (not currently supported on Ubuntu Linux).
- Network solution data such as SYZ-parameters

All of these models can be imported via the **Component Libraries** window by clicking **Models**.

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



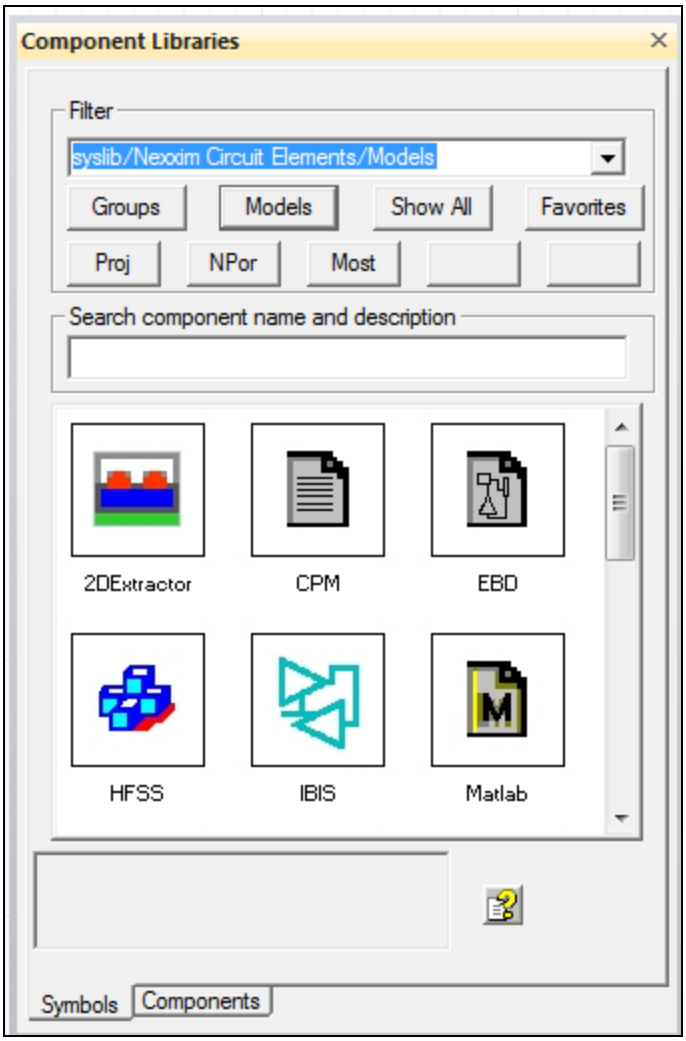
Double-click an icon matching the appropriate model type. Refer to the following topics.

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 Model to Import

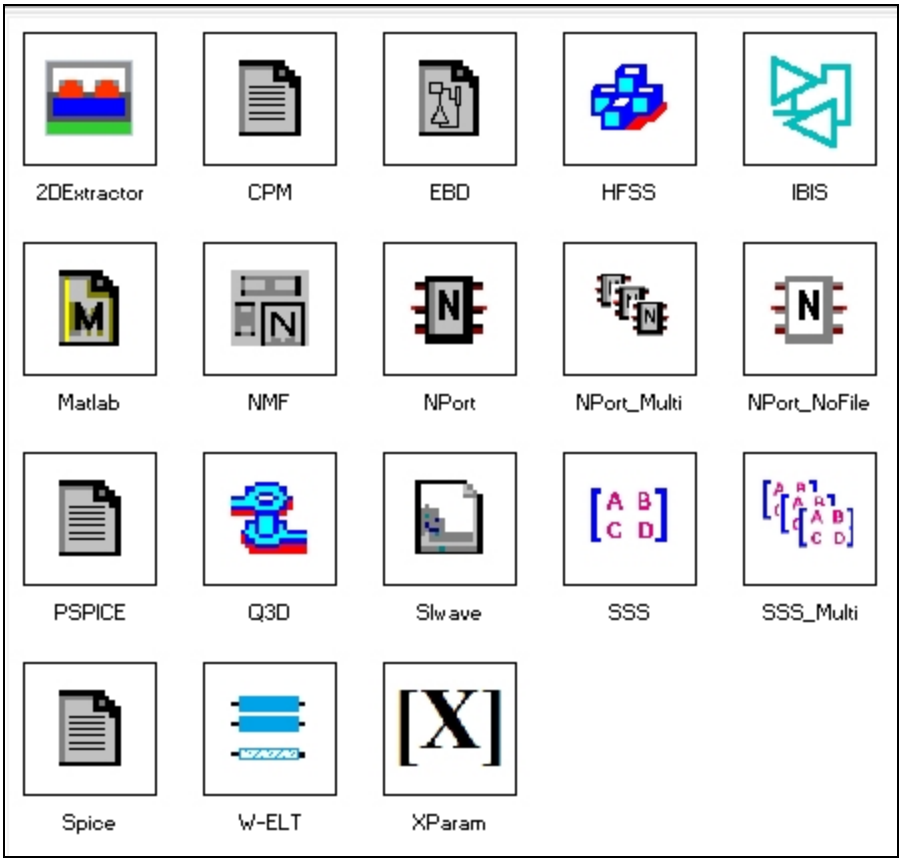
The Models Library is a library of Model Components that can be imported and placed in the schematic. Components such as NPorts, IBIS buffers, SPICE netlists, State-space models, W-elements, and X-Parameter elements are created from information in external files. Also use the Import Model facility to set up dynamic links to HFSS and other solver projects.

You can invoke the Models Library on the **Symbols** tab, by clicking **Models**.



The **Symbols** area displays the Model components.

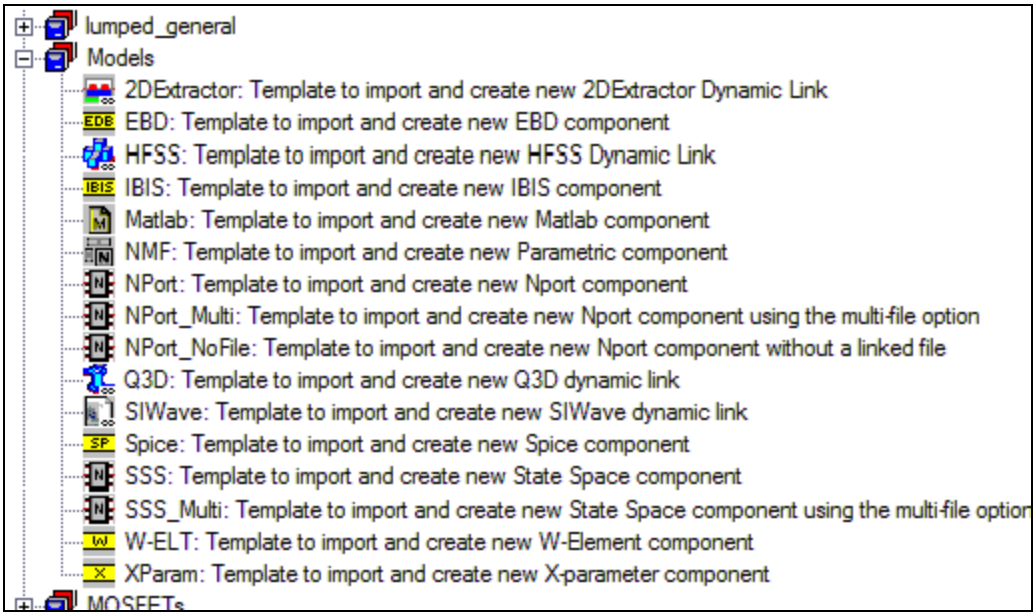
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clicking on any of these opens a window to select the file. When the file is open, if applicable, a second window opens to retrieve additional information. Once you click OK, clicking in the schematic places the new component.

These import items can also be accessed on the Component Libraries docking window like any component library. From the Components tab, expand the Models library to display the component types.

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



- Double-clicking any of the individual components opens a window to select a file or project of the selected type.
- Once a component of a given type has been created, you can filter for that type to see them all (e.g., after placing several NPort components, view the Symbols tab, and type "NP" into the **Filter** area and select "type: NPort". This filters to show only NPort components).

For more information, see Using the Component Libraries Window.

SPICE Library Support

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform allows you to import SPICE® libraries and SPICE components with an import wizard. Imported SPICE components can then be exported to AEDT library. The SPICE import window includes a check box for PSPICE models. A separate icon allows you to import a PSPICE model directly

> **Note:**
>
> Imported SPICE models and netlists must be in HSPICE, PSPICE, or Spectre format for simulation and post-processing with the Nexxim simulator.
>
> The Spice Import Wizard supports the following models:
>
> - amp, c, csw, d, l, ltra, njf, nmf, nmos, npn, pjf, pmf, pmos, pnp, r, sp, sw, u, urc, w

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

SPICE Library Import Procedure

To import a SPICE library or one of its components in a Circuit schematic:

1. View the **Component Libraries**. Open the **Symbols** area. Click **Models**.
2. From the area, select the **Spice** icon to open the **Select Model File** window.
3. The **Files of type** menu lists the file types for the SPICE file. You can import a SPICE library (.lib), a SPICE subcircuit definition (.cir, .spc, .sp), or a Spectre-format (.scs). You can specify the location of the SPICE file by clicking **Use Path**, **PersonalLib**, **UserLib**, or **SysLib**.

   > **Note:**
   >
   > **Import Spectre**. Spectre-format files are case-sensitive.

   Within the **PersonalLib**, **userlib**, or **syslib** folder, select the file from those listed, or type the name of the file in the **File Name** area. Note that **syslib** is reserved for libraries supplied with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. See Paths for Model File Selection for details.
4. Click **Open** in the **Select Model File** window to open the **Component Import** window.

The **Component Import** window lists all the SPICE components that are contained in the library.

- Check a box in the **Create Component** column to select the corresponding component for import.
- Click the **Symbol** and **Footprint** column boxes to assign a symbol and footprint to a component. These steps are optional; if either one is left unspecified, a default symbol or footprint is created automatically.
- Select **OK** to import the selected components and thereby display their names in the **Project Manager** window.

Once the component has been placed in the schematic, its Property Window will show a checkbox property named **NetlistAsSubcircuit**. Enabling the checkbox affects only what is written to the netlist for simulation. When the checkbox is enabled, the Spice model component will be netlisted inside its own subcircuit along with its file reference. This means the model name (taken from the Spice file) cannot conflict with the same model name in a different Spice file netlisted for a different component. If two Spice files with identical model names are read into the global scope, only data from one of the spice files will be used for all components referencing that model name, causing a simulation error. Checking the ”NetlistAsSubcircuit” property will move the file out of the global scope and prevent this problem.

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 the SPICE file has more than 10k characters in its DATA member, the parameter definitions from that file are not imported. The simulators gets the correct information on the parameters on the .lib or .sp file, but the parameters cannot be overridden on a per-instance basis.

Grouped Components

From the **Component Import** window, select **Group similar components** in order to group components that have the same parameter names and values. **Require consistent parameter values** (enabled only when **Group similar components** is selected) enforces that every element in the group has the same parameter values, which is the default.

The criteria for grouped components are:

- Components must be selected for import.
- Components must have the same number of pins.
- Models must have the same base model type (cap, bjt, etc).

After selecting **Group similar components** in the **Component Import** window, similar components are grouped in the display:

- After grouping, the **Name** column displays the file name of the component group appended with a group number; you can change this name by typing in the **Name** column.
- Multiple groups can be found, but every element of a group must have the same grouping criteria.
- A group can only be created if it contains multiple components.

Select **OK** to import the component group and thereby display its name in the **Project Manager** window.

- After placing the component group in the schematic, the component possesses a special property, **VComp**, that is displayed in the **Properties** window. Click the **VComp** property to select which model in the group are used in the schematic.
- Click **Choose Model** in the **Properties** window to open a **Model List** window that displays all the models in the group.
- You can then select a new model, click **OK,** and the new model name are reflected in the **Properties** window.

5. If you do not select your own symbol for an imported component, a default symbol are automatically selected. To select your own symbol, click **Symbol** to open the **Select Definition — Symbols** window. For more information, see Using the Symbol Editor.

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 you do not select your own footprint for an imported component, a default footprint are automatically selected. To select your own footprint, click **Footprint**. If you have encrypted libraries, this opens the **Enter Password** window.

After entering a valid password, or if you do not have encrypted libraries, the next window to open is the **Select Definition — Footprints** window. For more information, see Using the Footprint Editor.

7. If you select **Check this box if this is a PSPICE component** on the Component Import window, the component is imported as a PSPICE component.

The global tab of the netlist for the component includes lines (shown in **bold**) to switch to the PSPICE parser for this library, then back to Nexxim for the rest of the netlist:

```
simulator lang=pspice
.inc "C:\Documents\Ansoft\PersonalLib\Component Data
Files\Component Data Files\Spice\BC8\\
57_PNP_Model.lib"
simulator lang=nexxim
```

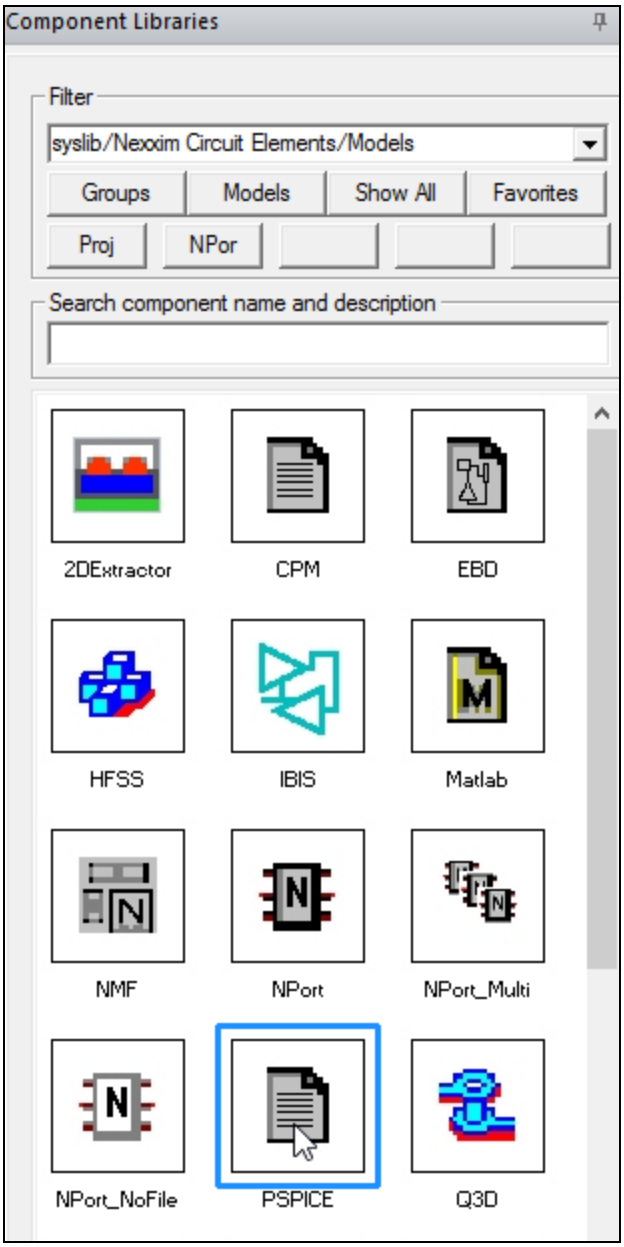
For more information about AEDT libraries see Using the Component Editor.

PSPICE Library Import Procedure

To import a PSPICE library or one of its components in a Circuit schematic:

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

1. View the **Component Libraries**. Open the **Symbols** area. Click **Models**.



2. From the area of icons, select the **PSPICE** icon to open the **Select Model File** window.
3. The **Files of type** menu lists the file types for the PSPICE file. You can import a PSPICE library (.lib) or a PSPICE subcircuit definition (.cir, .spc, .sp). You can specify the location of the PSPICE file by clicking **Use Path**, **PersonalLib**, **UserLib**, or **SysLib**.

Within the **PersonalLib**, **userlib**, or **syslib** folder, select the file from those listed, or type the name of the file in the **File Name** area. Note that **syslib** is reserved for libraries supplied with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. See Paths for Model File Selection for details.

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 **Open** in the **Select Model File** window.

The window lists all the SPICE components that are contained in the library.

- Check a box in the **Create Component** column to import the corresponding component.
- Click the **Symbol** and **Footprint** column boxes to assign a symbol and footprint to a component. These steps are optional; if either one is left unspecified, a default symbol or footprint is created automatically.
- Click **OK**.
- click the **Select Component** window to select one or more library components.
- Select **OK** to import the selected components and display their names in the **Project Manager** window.

The global tab of the netlist for the component includes lines (shown in **bold**) to switch to the PSPICE parser for this library, then back to Nexxim for the rest of the netlist:

```
simulator lang=pspice
.inc "C:\Documents\Ansoft\PersonalLib\Component Data
Files\Component Data Files\PSpice\BC8\\
simulator lang=nexxim
```

Once the component has been placed in the schematic, its Property Window will show a checkbox property named **NetlistAsSubcircuit**. Enabling the checkbox affects only what is written to the netlist for simulation. When the checkbox is enabled, the Spice model component will be netlisted inside its own subcircuit along with its file reference. This means the model name (taken from the Spice file) cannot conflict with the same model name in a different Spice file netlisted for a different component. If two Spice files with identical model names are read into the global scope, only data from one of the spice files will be used for all components referencing that model name, causing a simulation error. Checking the ”NetlistAsSubcircuit” property will move the file out of the global scope and prevent this problem.

For more information about AEDT libraries see Using the Component Editor.

Modifications to SPICE Pin Names

When importing a SPICE component, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform may modify pin names for compatibility with AEDT pin names and guarantee uniqueness of pin names.

- The following characters are invalid in AEDT pin names and are removed (squeezed out) from pin names: []<>{}&*<space>,\
- No other modification to pin names are made. Capitalization at the location of removed characters is not adjusted.
- A uniqueness test is performed. Any non-unique names have _## appended to the name, where ## is a number that increments until a unique name is generated. Pin names are

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

processed in the order they are specified on the .subckt line. No check is performed to see if the name already has a _## in it and attempt to increment that number. Thus if there are two P_1 pins, the result is a P_1 pin and a P_1_1 pin.

PSPICE Model Support

The following types of models are supported:

- Semiconductor
- PSPICE Function
- PSPICE Vendor Libraries

Semiconductor Model Support

The following table shows the supported PSPICE semiconductor models.

Type	PSPICE	Support	Comments
D<name> - d	PSPICE diode	Supported	
M<name> - nmos/pmos	Level=1 Shichman-Hodges	Supported	
M<name> - nmos/pmos	Level=2 geometry-based, analytic	Available[2]	
M<name> - nmos/pmos	Level=3 semi-empirical, short-channel	Supported	
M<name> - nmos/pmos	Level=4 BSIM	Available[2]	Level=13
M<name> - nmos/pmos	Level=5 EKV version 2.6	Available[2]	Level=55
M<name> - nmos/pmos	Level=6 BSIM3 version 2.0	Not supported	
M<name> - nmos/pmos	Level=7 BSIM3 version 3.2	Supported	
J<name> - njf/pjf	PSPICE Junction FET	Supported	
B<name>, gasfet	Level=1 Curtice	Available[2]	NJF Level=3 SAT=0
B<name>, gasfet	Level=2 Raytheon or Statz	Available[2]	NJF Level=3 SAT=2 or Level=25
B<name>, gasfet	Level=3 TOM by Triquint	Available[2]	NJF Level=3 SAT=3
B<name>, gasfet	Level=4 Parker-Skellern	Available[2]	Level=18
B<name>, gasfet	Level=5 TOM-2 by Triquint	Not supported	
B<name>, gasfet	Level=5 Tom-3 by Triquint	Available[2]	Level=7

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

Type	PSPICE	Support	Comments
Q<name> - npn/pnp	Level=1 Gummel-Poon	Supported	
Q<name> - npn/pnp	Level=2 Mextram	Available[2]	
Z<name> - nigbt		Not supported	Not implemented in simulator

Note:

1. For MESFETs, PSPICE uses B<name> for instance and GASFET to call the model. Nexxim uses J<name> and NJF/PJF.
2. "Available" in the table means the model is available but has not been tested for PSPICE compatibility

PSPICE Function Support

The following table lists the built-in functions available for use in parameter statements, user-defined functions, and expression-based behavioral circuit elements.

Function	Support in Nexxim
Built in Functions	
ABS(x)	Supported
ACOS(x)	Supported
ACOSH(x)	Supported
ARCTAN(x)	Supported
ASIN(x)	Supported
ASINH(x)	Supported
ATAN(x)	Supported
ATAN2(y,x)	Supported
ATANH(x)	Supported
COS(x)	Supported
COSH(x)	Supported

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

DDT(x)	Supported
EXP(x)	Supported
IF(t, x, y)	Supported
IMG(x)	Not Supported
LIMIT(x,min,max)	Supported
LOG(x)	Supported
LOG10(x)	Supported
M(x)	Supported
MAX(x,y)	Supported
MIN(x,y)	Supported
P(x)	Not Supported
PWR(x,y)	Supported
PWRS(x,y)	Supported
R(x)	Not Supported
SCHEDULE(x1,y1,x2,y2,... xn,yn)	Not Supported
SDT(x)	Not Supported
SGN(x)	Supported
SIN(x)	Supported
SINH(x)	Supported
STP(x)	Supported
SQRT(x)	Supported
TAN(x)	Supported
TANH(x)	Supported
TABLE(x,x1,y1,x2,y2,... xn,yn)	Supported

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

ZERO(expression)	Not Supported
ONE(expression)	Not Supported
CEIL(arg)	Supported
FLOOR(arg)	Supported
INTQ(arg)	Not Supported
User-Defined Functions	
.FUNC (function)	Supported

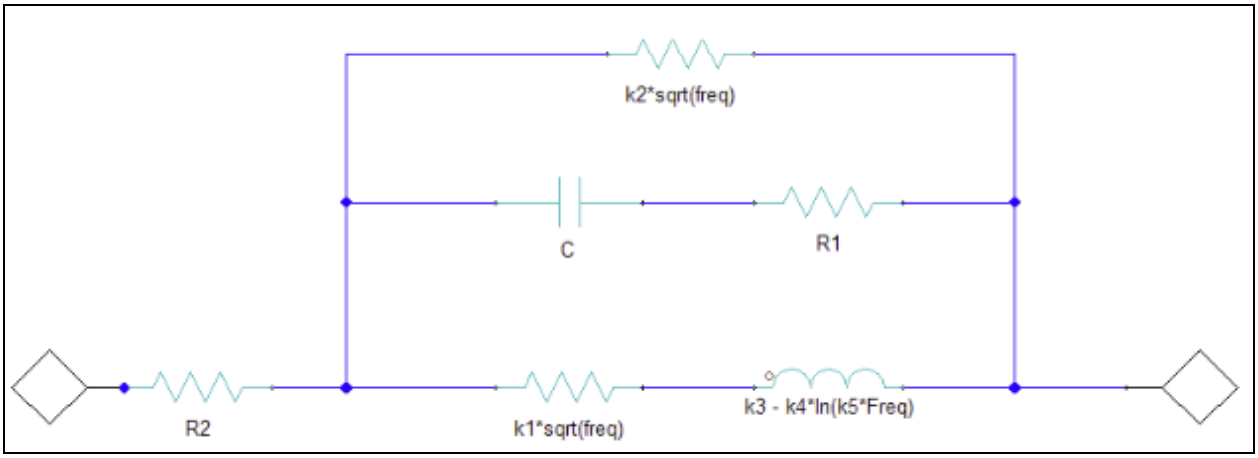
PSPICE Vendor Library Support

Coilcraft Power Inductors are supported.

Coilcraft Power Inductors

The following lumped-element spice model on the manufacturer is implemented for Power Inductor parts. Model parameters for the elements are those specified in the datasheet tables for the associated parts.

Time-domain simulation is enabled by evaluating all frequency-dependent equations at 100KHz. This frequency can be adjusted by modifying the parameter ‘tran_eval_freq’ in the model library.



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

Coilcraft Power Inductors (Series Name)		
0603ps	LPS3008-561	MSS6122
0805PS	LPS3010	MSS6132
1008PS	LPS3015	MSS7341
1812PS	LPS3314	mvr
DO1605	LPS4012	PFL1005
DO1606	LPS4018	PFL1610
DO1608	LPS5010	pfl2010
DO1813H	LPS5015	PFL2510
DO3308P	LPS5030	PFL2512
DO3314	LPS6225	PFL4514
DO3316HC	LPS6235	PFL4517
DO3316P	LPZ3008	SER1360
DO3316T	me3215	SER1590
DO3340P	me3220	SER2009
DO5010H	MLC1265	slc1175
DO5022P	MOS6020	slc7649
EPL2010	MSS1038	slr1050
EPL2014	MSS1038T	slr1065
EPL3012	MSS1048	slr1070
EPL3015	MSS1048T	slr1075
GA3199	MSS1210	slr1190
HA3778	MSS1246	xal1350

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

Coilcraft Power Inductors (Series Name)		
LPO1704	MSS1246T	xal40xx
LPO2506	mss1260	xal50xx
LPO3310	MSS1260T	xal60xx
LPO4812	MSS1278	xal7020
LPO4815	MSS1278T	xal7030
LPO6013	MSS4020	xel3515
LPO6610	MSS5131	

IBIS Library Support

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform provides an easy way to import IBIS® buffer components, pins and models from IBIS libraries and place them as components in the schematic. Property settings allow you to set up IBIS transmitters, receivers, and other elements, including IBIS-AMI components.

To import and use an IBIS buffer component, use the following steps:

1. Import IBIS Library File
2. Import IBIS Single Component or Pin or Buffer Element
3. Place IBIS Buffer Component in Schematic
4. Edit IBIS Component Properties
5. IBIS File Import Technical Notes

A separate procedure allows the import of IBIS EBD Files:

- Importing IBIS EBD Files

An IBIS I/O Wizard is available for connecting schematic signal, power, and ground nodes to IBIS buffer instances:

- IBIS I/O Wizard

A component defined by the IBIS Algorithmic Model Interface (AMI) specification requires special import and setup steps, described in the AMI Analysis topic.

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 HSPICE users)

AEDT IBIS Components created with the "Import IBIS Components" procedure contain certain convenience features that are only tested for compatibility with the Nexxim simulator.

1. **Pin Imports,** which provide automatic instantiation of the lumped RLC package parasitics on the **[Package]** or **[Pin]** keywords.
2. The **buffer_mode** parameter **Internal** settings, which allow users to specify the mode of the device (Input, Output, High Impedance) from a pull down list instead of creating a voltage source to drive the enable node.
3. The **logic_in** parameter **Internal** setting, which allows users to configure a built-in eye source instead of creating an external voltage source to provide the bit pattern.

Follow these steps when importing IBIS components to create a Circuit Design that is compatible with other simulators.

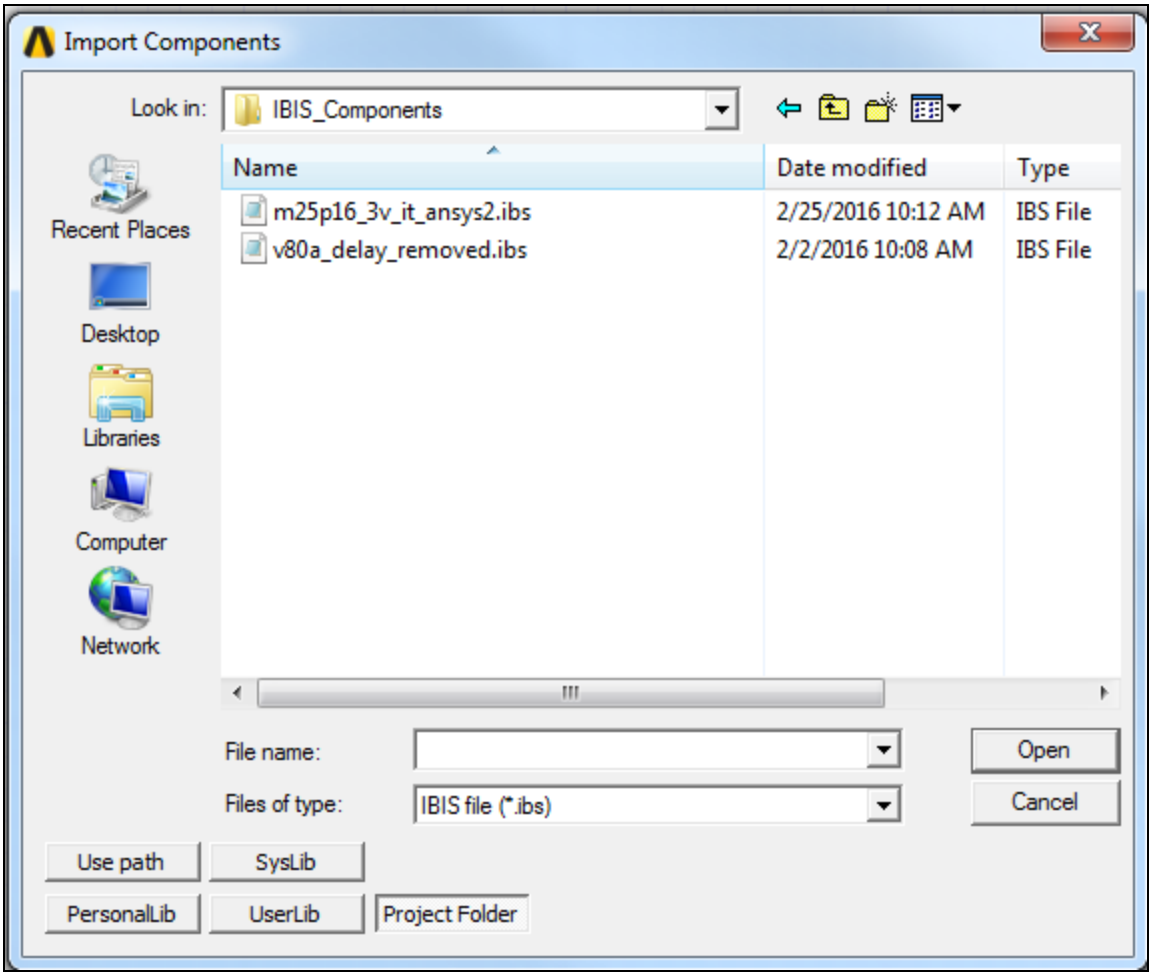
1. Click the **Buffer Import** tab in the **Import IBIS Components** window. Do not click the **Pin Import** tab.
2. When placing the components on the schematic and configuring the circuit, provide external sources and click the **Select Net** option to connect the **buffer_mode** (enable) and **logic_in** (bit source) parameters to the external sources.

Import IBIS Library File

To import an IBIS buffer or buffer model from an IBIS library file (.ibs) in a Circuit schematic:

1. View the **Component Libraries**. Open the **Symbols** area. Click **Import Models**. From the area, select the **IBIS** icon to open the **Import Components** 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. Specify the location of the IBIS file by clicking **Use Path**, **PersonalLib**, **UserLib**, or **SysLib**.

See Paths for Model File Selection for details.

3. Within the **PersonalLib**, **userlib**, or **syslib** folder, select the file from those listed, or type the name of the file in the **File Name** area. Note that **syslib** is reserved for libraries supplied with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform.

If you select **Use path**, type the name of the file in the **File Name** area, or click the **Look in** field to navigate to the file and record its name. Note that components or libraries imported with Use Path may not be portable when the project is moved to another machine.

When **Project Folder** is selected, references to the file in the design are relative to the directory where the project resides. In this case, the path is saved in the project **.adsn** file as a variable 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.

```
.lib '$PROJECTDIR/x_113854.ibs'
```

The variable $PROJECTDIR are expanded to the current location of the project when the design is converted to a netlist and run by an analysis tool. Otherwise, an absolute path is saved. If you move a project and its library files together to a new directory, you can preserve the file references by selecting the **Project Folder** option.

4. Click **Open** to select the IBIS file and open the **Import IBIS** window. The next step is to select the pins and buffer models to make into schematic components.

Import IBIS Single Component or Pin or Buffer Element

Import IBIS Single Component or Pin or Buffer Element

IBIS devices are created in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform using Single Component Import, Pin Import or Buffer Model Import. A component import can include [Package Model] and [Pin Mapping] information to control power and parasitic package behavior. A Pin Import captures the behavior of a given pin on a component defined in the IBIS file. A device created via Pin Import contains the behavior of the model associated with the pin, and it also contains information about the parasitic package model specified for that pin. A buffer model import captures the behavior of a given model in an IBIS file without any package parasitic modeling.

The **Import IBIS** window enables the following actions:

- Import a complete component with a [Package Model] and importing all signal, power, ground, and NC pins. See Import Single Component with [Package Model], All Pins.
- Import a complete component with a [Package Model], but importing only selected signal, power, ground, and NC pins. See Import Single Component with [Package Model], Selected Pins.
- Import a complete component without a [Package Model], importing all signal pins. See Import Single Component with No [Package Model], All Pins.
- Import a complete component without a [Package Model], importing only selected signal pins. See Import Single Component with No [Package Model], Selected Pins.
- Import selected pins from a selected component. See Pin Import Tab.
- Import a buffer model without referencing a specific component or pin. See Buffer Import Tab.

Import Single Component with [Package Model], All Pins

An IBIS buffer imported as a single component has a rectangular schematic symbol with terminals for the imported pins in the IBIS model. When no individual pins are selected, Nexxim imports all eligible pins. When the IBIS component definition includes a [Package Model], and **Use Package Model** is selected, Nexxim uses the information in the [Package Model] to instantiate RLC parasitics between the die-side and the board-side nodes. The power and

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

ground pins become available for import, and the **Component External Power** property is present. Then you can connect the pins to signals, power sources, and grounds in the schematic. The **Component External Power** property is set to ON, so the component expects power through the component terminals.

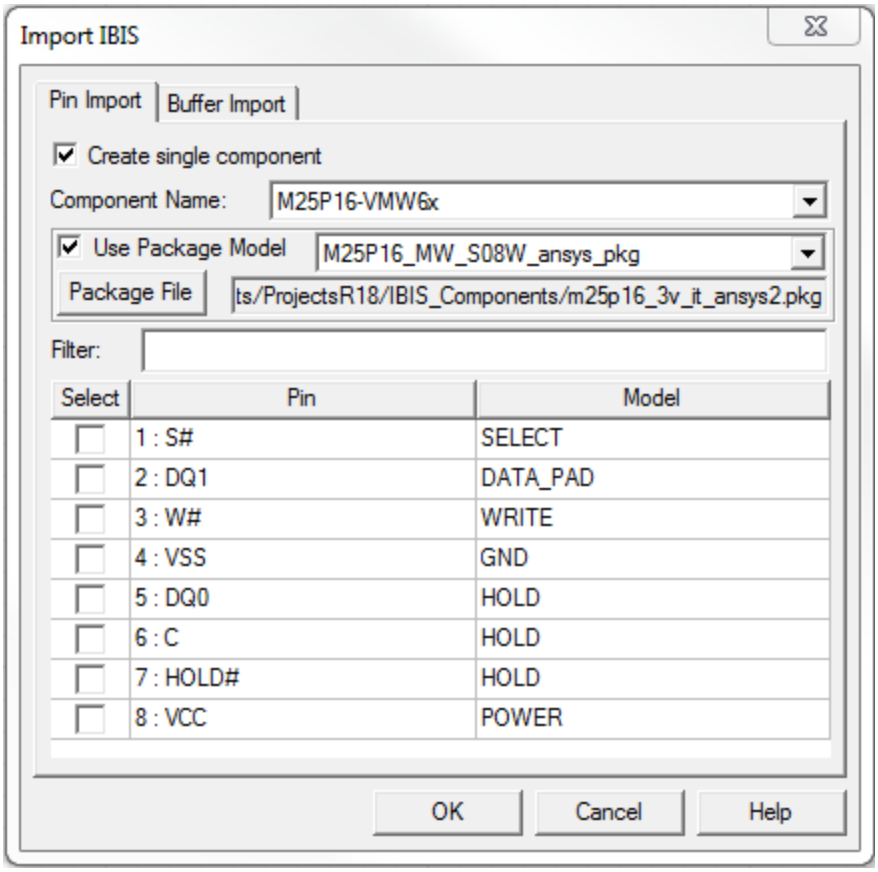
When the component definition includes a [Pin Mapping], Nexxim constructs the specified power and ground buses internal to the component. A [Pin Mapping] is required in all cases where the component has multiple POWER and GND buses. However, Nexxim can make the power bus connections without a [Pin Mapping] when there is exactly one POWER bus and one GND bus, identified by their IBIS signal_name entries. Each bus (signal_name) can have multiple pins, which are tied together. Nexxim halts with an error message if there is no [Pin Mapping] and the component does not have exactly one POWER bus and one GND bus.

When you have identified the IBIS library file in the **Import Components** window, the **Import IBIS** window appears.

To import a single IBIS component that includes a [Package Model] and importing all component signal, power, ground, and no-connect (NC) pins:

1. Click the **Create single component** check box at the top of the **Pin Import** tab.
2. From the **Component Name** field, use the drop-down menu to select the component with a [Package Model] on the IBIS file.
3. If the component definition includes a [Package Model], click the **Use Package Model** check box and choose the package model from the drop-down menu.
4. If the [Package Model] is defined in a separate .pkg file rather than in the IBIS file, click **Package File** and select the package model file.
5. Do not select any individual pins. With no individual pins selected, Nexxim imports ALL the signal, POWER, GND, and NC pins on the component definition.
6. Click **OK** to create the component and add it to the **Definitions** > **Components** listing 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.



(See *Power Nodes* in the "IBIS File Import Technical Notes" on page 5-142 for details on the methods for supplying power to imported IBIS components. See *Package RLC Parasitics* in the technical notes for details on support for RLC parasitics in IBIS components.

The next step is to place the IBIS components in the schematic.

Place IBIS Buffer Component in Schematic

Import Single Component with [Package Model], Selected Pins

An IBIS buffer imported as a single component has a rectangular schematic symbol with terminals for the imported pins in the IBIS model. When individual pins have been selected, only those pins are imported. When the IBIS component definition includes a [Package Model], and **Use Package Model** is selected, Nexxim uses the information in the [Package Model] to instantiate RLC parasitics between the die-side and the board-side nodes. The power and ground pins become available for import, and the **Component External Power** property is present. Then you can connect the pins to the individual signals, power sources, and grounds in the schematic. The **Component External Power** property is set to ON, so the component expects power through the component terminals. When the component definition includes a [Pin Mapping], Nexxim constructs the power and ground buses internal 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.

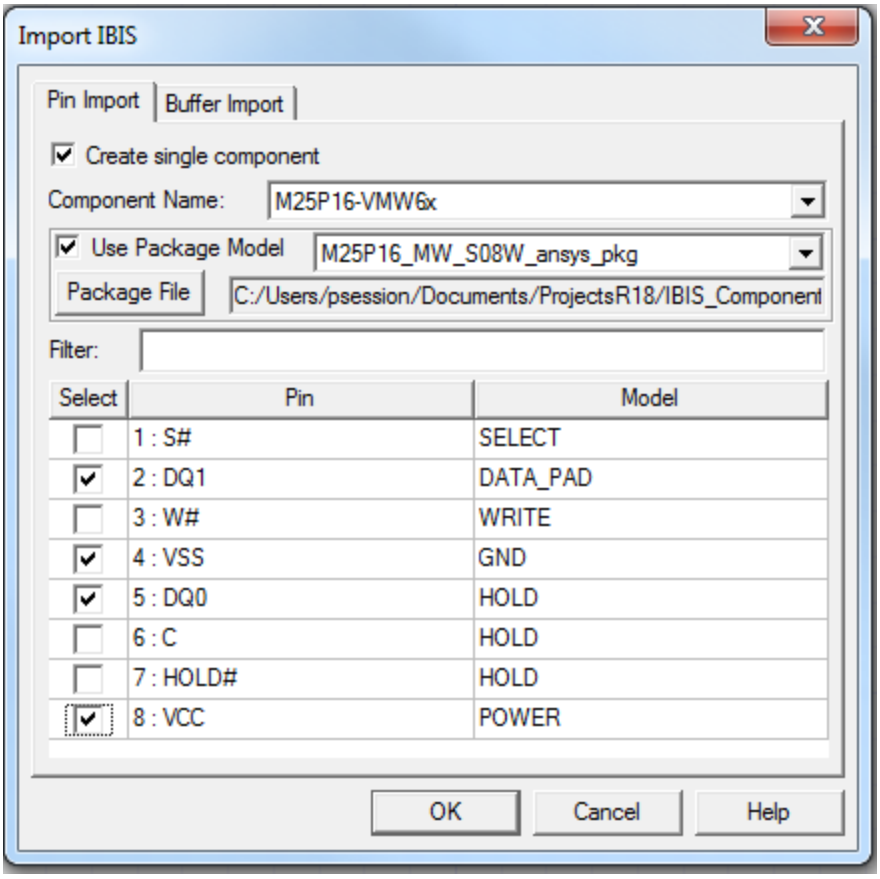
The pins selected for import must include a subset of POWER and GND pins sufficient to power any power rail (bus) used by any of the imported signal pins (e.g., if one of the imported signal pins specifies a bus "VCC" in the pu_ref column of its [Pin Mapping] entry, and no power pin attached to the "VCC" bus is imported, Nexxim issues an error).

When you have identified the IBIS library file in the **Import Components** window, the **Import IBIS** window appears.

To import a single IBIS component that includes a [Package Model] with selected component signal, power, ground, and no-connect (NC) pins:

1. Click the **Create single component** check box at the top of the **Pin Import** tab.
2. From the **Component Name** field, use the drop-down menu to select the component with a [Package Model] on the IBIS file.
3. If the component definition includes a [Package Model], click the **Use Package Model** check box and choose the package model from the drop-down menu.
4. If the [Package Model] is defined in a separate .pkg file rather than in the IBIS file, click **Package File** and select the package model file.
5. Select the appropriate individual pins. Nexxim imports only the selected signal, POWER, GND, and NC pins.
6. Click **OK** to create the component and add it to the **Definitions** > **Components** listing 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.



(See *Power Nodes* in the "IBIS File Import Technical Notes" on page 5-142 for details on the methods for supplying power to imported IBIS components. See *Package RLC Parasitics* in the technical notes for details on support for RLC parasitics in IBIS components. The next step is to place the IBIS components in the schematic.

Place IBIS Buffer Component in Schematic

Import Single Component with No [Package Model], All Pins

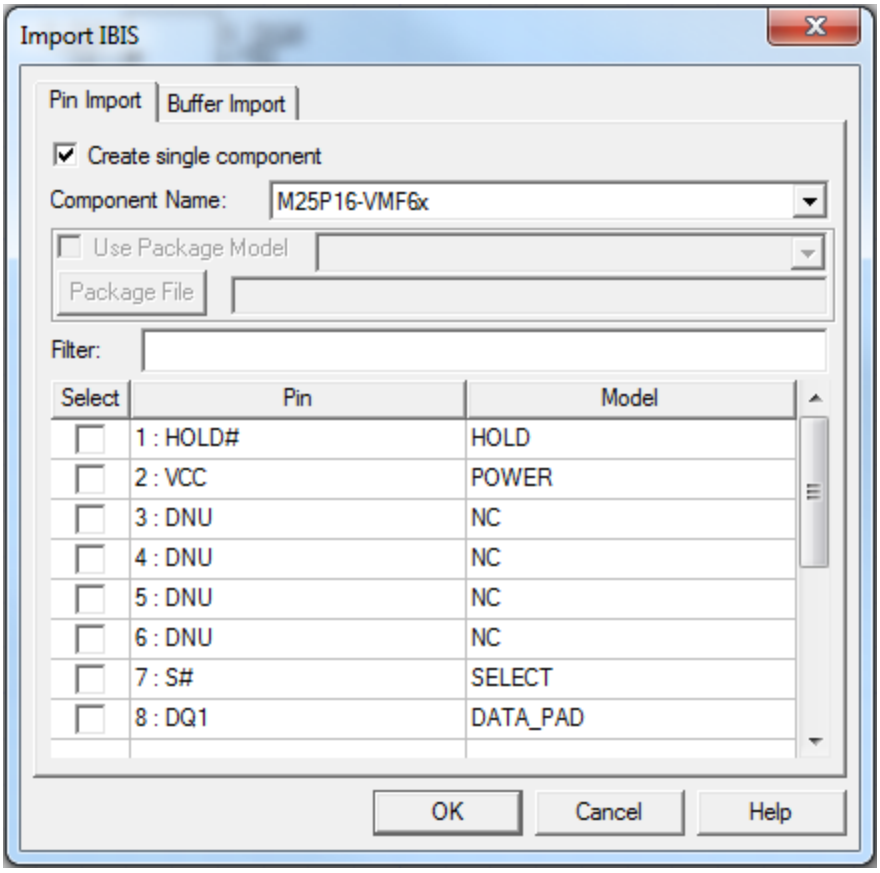
An IBIS buffer imported as a single component has a rectangular schematic symbol with terminals for the imported pins in the IBIS model. When no individual pins are selected, Nexxim imports all eligible pins. When the IBIS component definition does not include a [Package Model], the [Package Model] option is deactivated, and the power and ground pins are not available for import. Then you can connect the pins to signals in the schematic. The component expects power as specified by the **power** parameter on each pin. Any [Pin Mapping] in the component definition is ignored. Nexxim uses the [Pin] or [Package] data (selected by **pkg_ selector** property) to instantiate RLC parasitics between the die-side and the board-side nodes.

When you have identified the IBIS library file in the **Import Components** window, the **Import IBIS** 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.

To import a single IBIS component without a [Package Model] and importing all component signal pins:

1. Click the **Create single component** check box at the top of the **Pin Import** tab.
2. From the **Component Name** field, use the drop-down menu to select the component on the IBIS file.
3. Do not select any individual pins. With no individual pins selected, Nexxim imports ALL the signal pins on the component definition.
4. Click **OK** to create the component and add it to the **Definitions** > **Components** listing in the **Project Tree**.



(See *Power Nodes* in the "IBIS File Import Technical Notes" on page 5-142 for details on the methods for supplying power to imported IBIS components. See *Package RLC Parasitics* in the technical notes for details on support for RLC parasitics in IBIS components. The next step is to place the IBIS components in the schematic.

Place IBIS Buffer Component in Schematic

Import Single Component with No [Package Model], Selected Pins

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

An IBIS buffer imported as a single component has a rectangular schematic symbol with terminals for the imported pins in the IBIS model. When individual pins have been selected, only those pins are imported. When the IBIS component definition does not include a [Package Model], the [Package Model] option is deactivated, and the power and ground pins are not available for import. Then you can connect the pins to the individual signals in the schematic. The component expects power as specified by the **power** parameter on each pin. Any [Pin Mapping] in the component definition is ignored. Nexxim uses the [Pin] or [Package] data (selected by **pkg_selector** property) to instantiate RLC parasitics between the die-side and the board-side nodes.

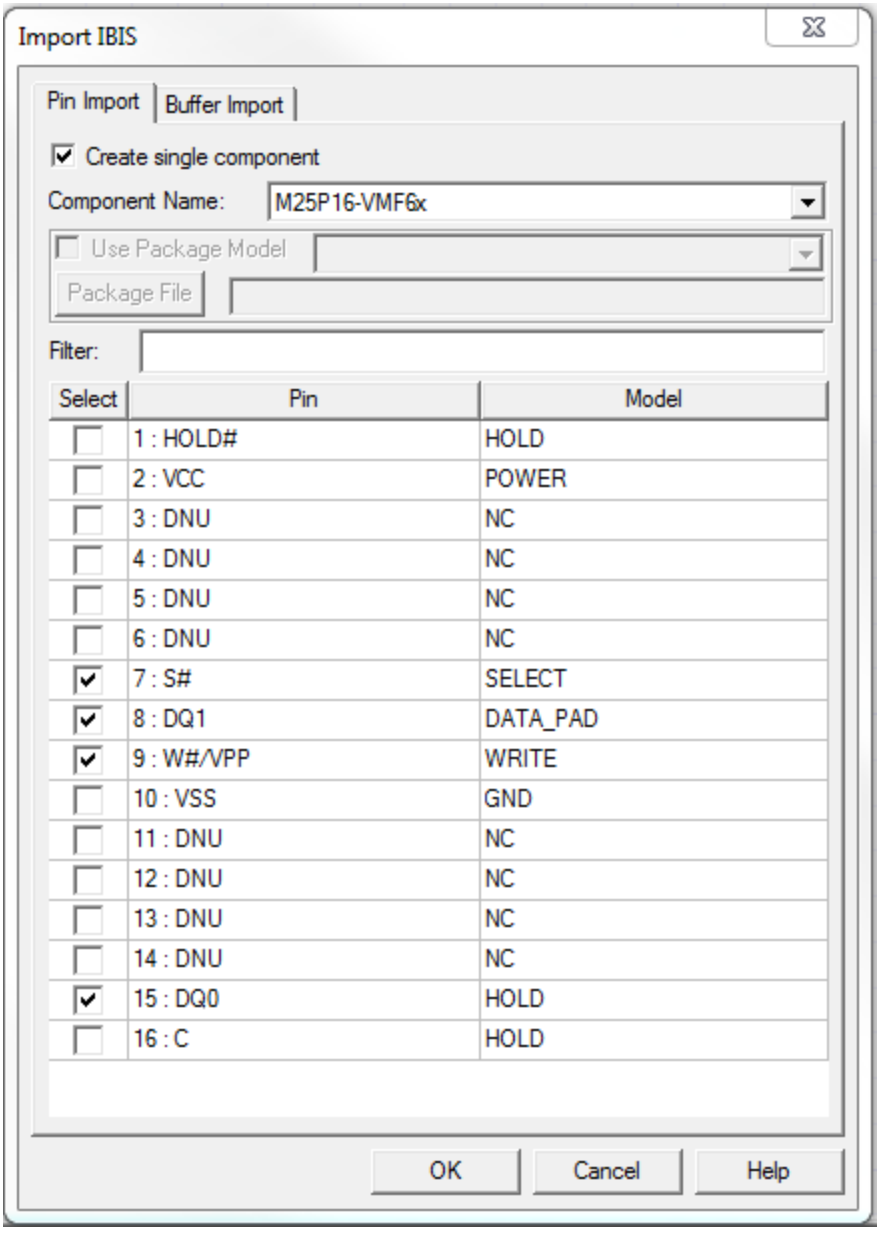
When you have identified the IBIS library file in the **Import Components** window, the **Import IBIS** window appears.

To import a single IBIS component without a [Package Model] with selected component signal pins:

1. Click the **Create single component** check box at the top of the **Pin Import** tab.
2. Select the appropriate individual pins. Nexxim imports only the selected signal pins.
3. Click **OK** to create the component and add it to the **Definitions** > **Components** listing 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 **Project Manager** window.



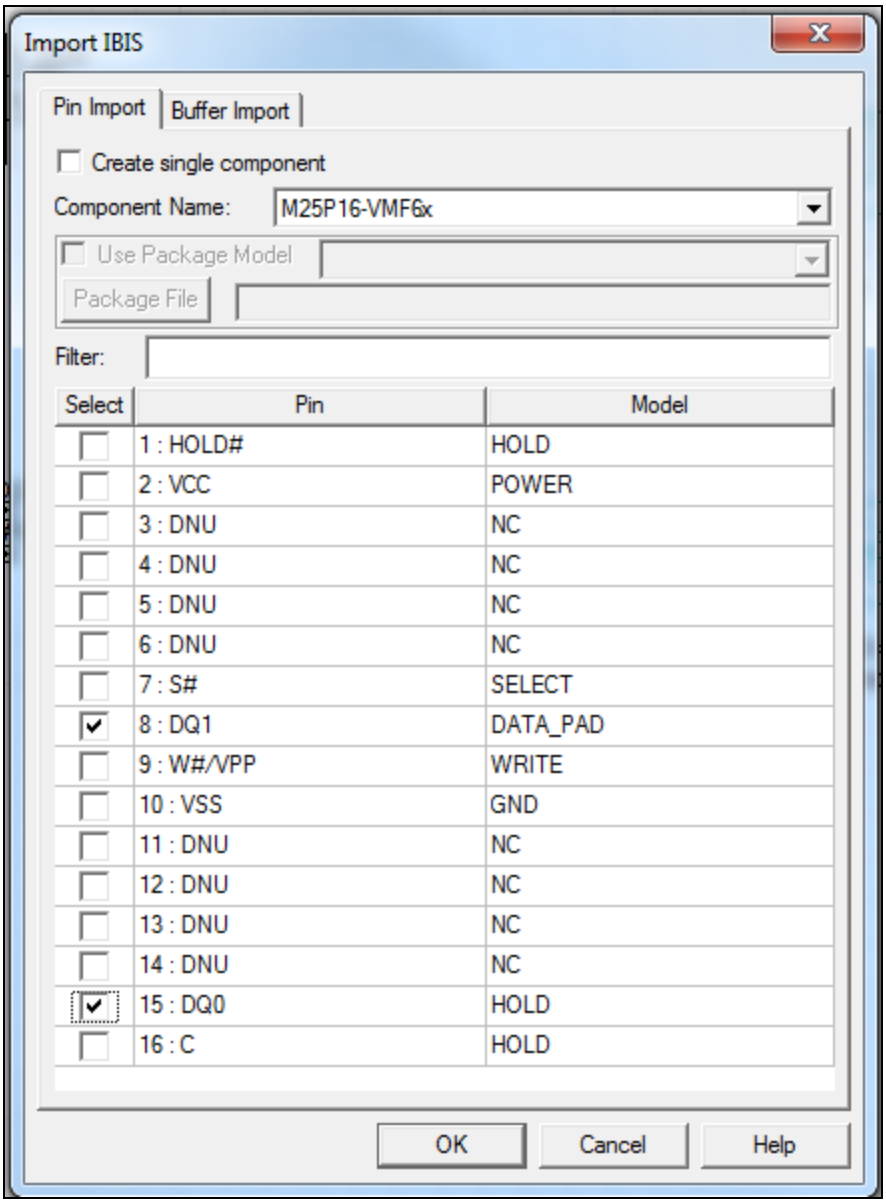
(See *Power Nodes* in the "IBIS File Import Technical Notes" on page 5-142for details on the methods for supplying power to imported IBIS components. See *Package RLC Parasitics* in the technical notes for details on support for RLC parasitics in IBIS components. The next step is to place the IBIS components in the schematic.

Place IBIS Buffer Component in Schematic

Pin Import Tab

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

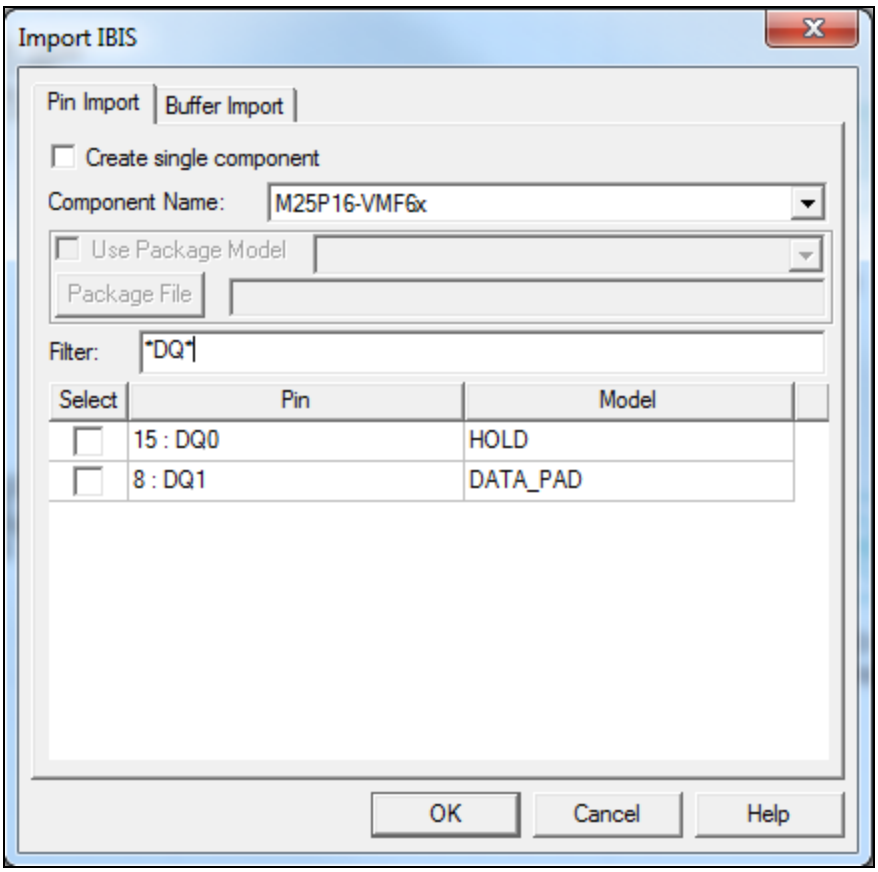
Click the **Pin Import** tab to import one or more buffer pins from a specified IBIS component. Each pin imported with **Pin Import** has a buffer model (or model selector). Each pin gets its power as specified in its **power** parameter, and uses the package model selected by a **Package Model** import or by the pin's individual **pkg_selector** parameter (which is not displayed when a **Package Model** was imported).



1. The **Component Name** field lists all the IBIS components contained in the selected IBIS library. Scroll the list and select the name of the component to import.

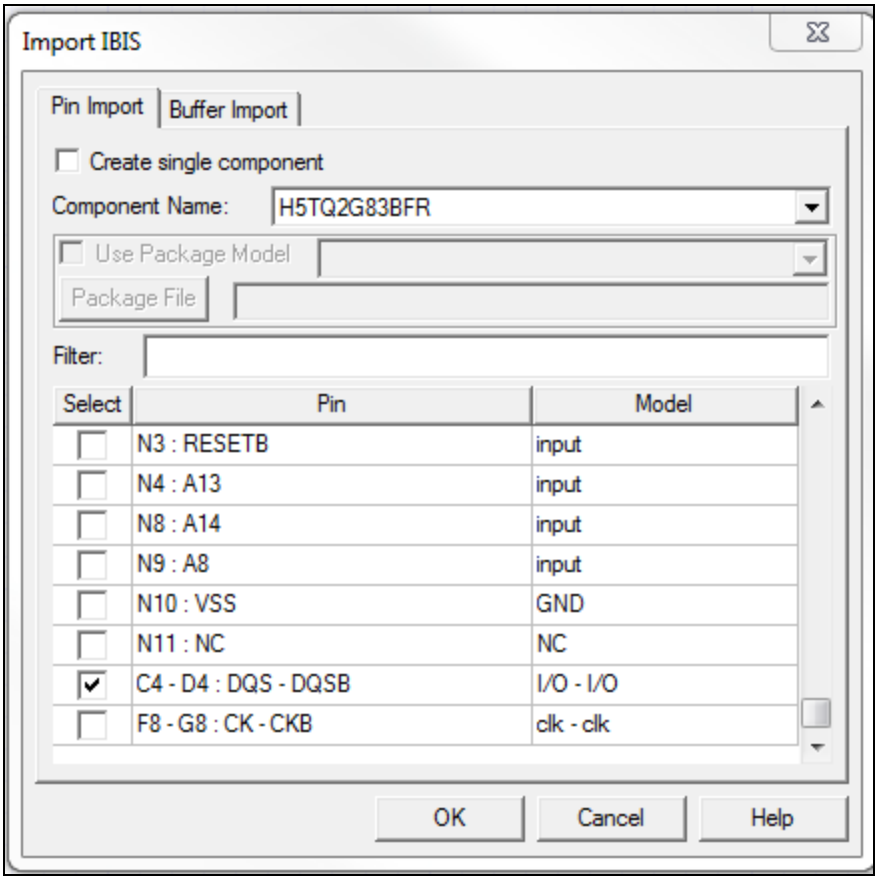
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. To import the pins as a single component where each pin retains its own model and buffer type, check **Create single component**. See Import IBIS Single Component or Pin or Buffer Element for details.
3. The **Pin Import** tab displays the pins defined for the component in the IBIS file, and the model or model selector associated with each pin. Single pins are listed first, then differential pin pairs if any are defined in the IBIS file.
4. To create a muiti-pin component:
    - Use the check boxes to select one or more pins to import.
    - Alternatively, click the **Filter**: field to enter a pin name, which can use asterisks as wild-card characters. Click **Select** at the top of the column to automatically select any pin names that match the Filter:



If the IBIS file contains differential pins, they appear in the pin list:

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



Here the two differential pins (C4-D4: DQS - DQSB) have been selected.

5. Click **OK** to generate the components on the IBIS Pin Import.

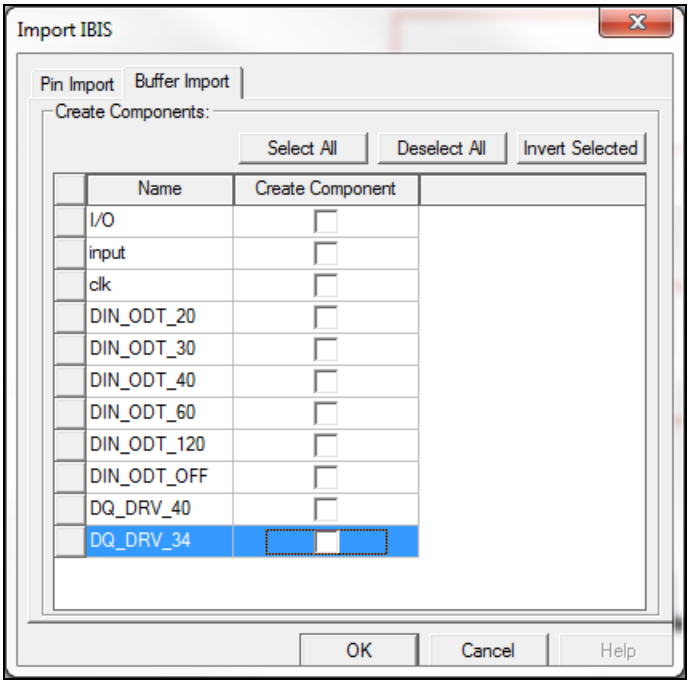
(See *Power Nodes* in the "IBIS File Import Technical Notes" on page 5-142 for details on the methods for supplying power to imported IBIS components. See *Package RLC Parasitics* in the technical notes for details on support for RLC parasitics in IBIS components. The next step is to place the IBIS components in the schematic.

Place IBIS Buffer Component in Schematic

Buffer Import Tab

The **Buffer Import** tab allows users to create an Ansys Electronics Desktop (AEDT)™ component representing a single buffer model or model selector from an IBIS file. Each pin gets its power as specified in its **power** parameter, and has no package model support.

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



(Refer to *Power Nodes* in the "IBIS File Import Technical Notes" on page 5-142 for details on the methods for supplying power to imported IBIS components. Refer to *Package RLC Parasitics* in the technical notes for details on support for RLC parasitics in IBIS components. The next step is to place the IBIS components in the schematic.

If users check the **Create Component** box for any of the imported buffer models, the next step is to place the IBIS components in the schematic.

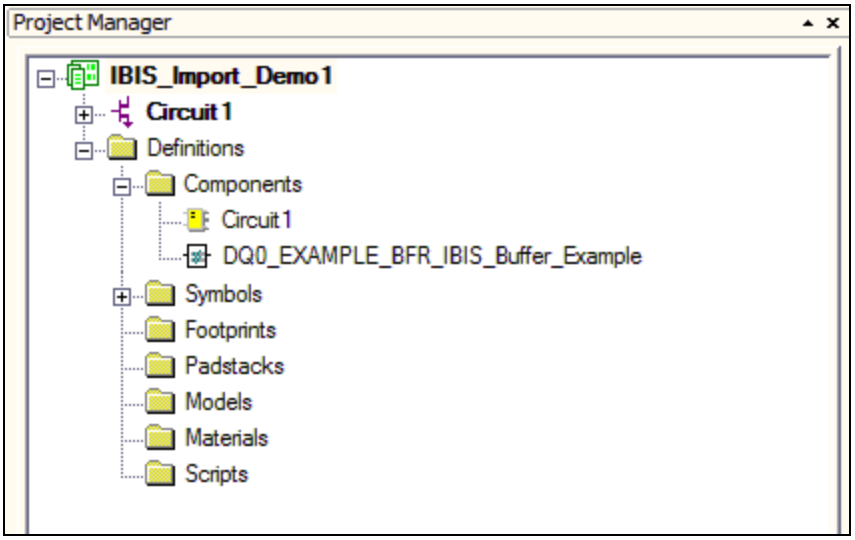
Place IBIS Buffer Component in Schematic

Place IBIS Buffer Component in Schematic

In most cases, after clicking OK on the **IBIS Import** window, a copy of the symbol for the component are attached to the cursor so you can place it in the schematic. Also generate an instance of an IBIS buffer component from the Definitions folder:

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

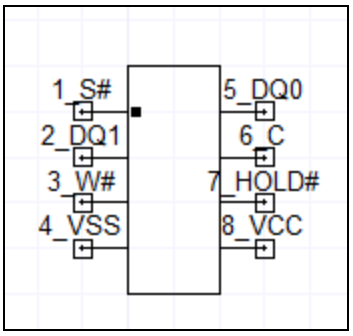
1. Expand the Definitions folder in the Project window, then expand the Components folder.



2. Click selected component and drag the symbol into the schematic.

Component Import, Package Model, All Pins Imported

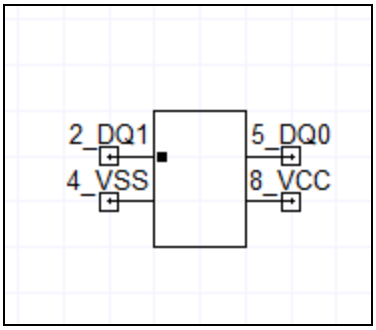
IBIS single components are named for the IBIS library file and the [Component] entry. The symbol shows all the pins with pin names and signal_names read from the file. This is the symbol for component with a [Package Model] and all pins imported. The symbol includes all signal, POWER, and GND pins for connection in the schematic.



Component Import, Package Model, Selected Pins Imported

IBIS single components are named for the IBIS library file and the [Component] entry. The symbol shows all the pins with pin names and signal_names read from the file. This is the symbol for component with a [Package Model] with some selected pins imported. The symbol includes only the selected signal, POWER, and GND pins for connection in the schematic.

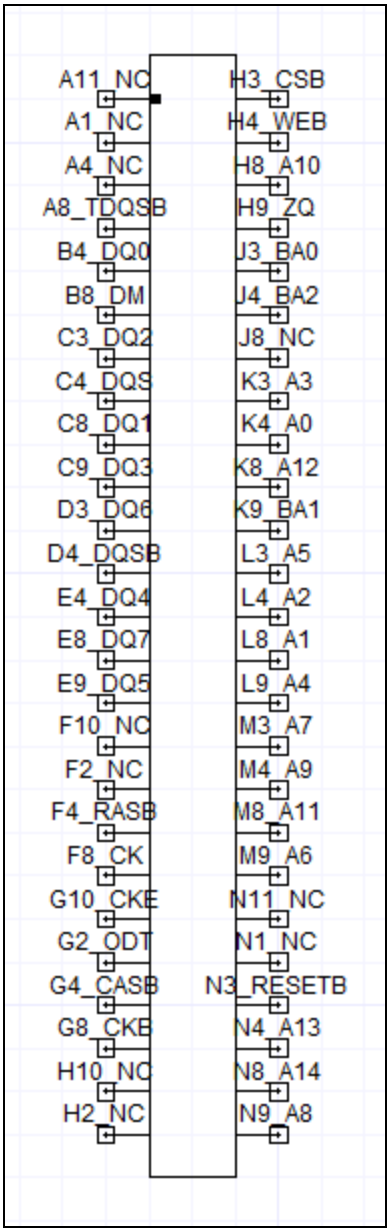
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Component Import, No Package Model, All Pins Imported

IBIS single components are named for the IBIS library file and the [Component] entry. The symbol shows all the pins with pin names and signal_names read from the file. This is the symbol for a component with no [Package Model] and all signal pins imported (no individual pins selected). The symbol includes all signal pins for connection in the schematic.

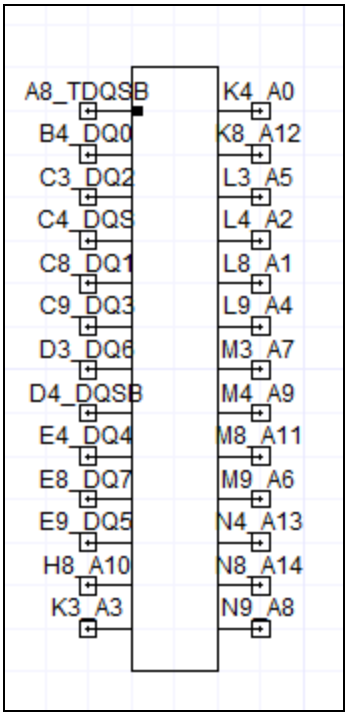
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Component Import, No Package Model, Selected Pins Imported

IBIS single components are named for the IBIS library file and the [Component] entry. The symbol shows all the pins with pin names and signal_names read from the file. This is the symbol for the component in the preceding screenshot with no [Package Model] and selected signal pins imported. The symbol includes just the selected signal pins for connection in the schematic.

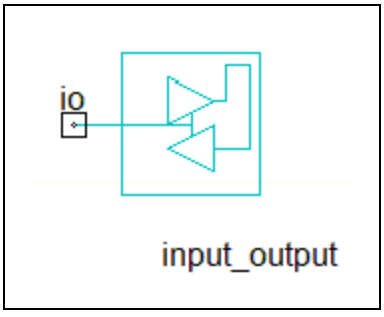
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Pin Import or Buffer Model Import

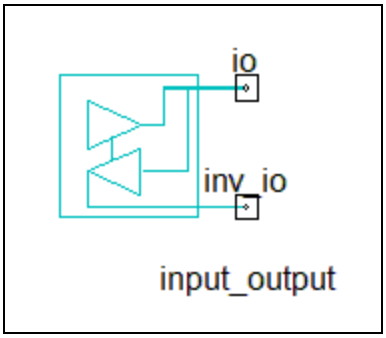
IBIS Pin Import components typically have names that begin with the pin name. IBIS Buffer Model Import component names begin with the model name or model selector name.

Single-pin IBIS components all use the following default schematic symbol. The text changes to show the buffer type selected for each component instance. The example shows the **input_ output** buffer type.



Differential IBIS components all use the following default schematic symbol.

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 are finished placing instances of the component, right-click and select **Finish** or press **Esc** on the keyboard to stop placing instances.

The last step in the import process is to verify the settings of the component properties.

Edit IBIS Component Properties

Edit IBIS Component Properties

Right-click the IBIS buffer symbol and select **Properties** to open the **Property** window. The available properties depend on the buffer type and model. For imported Pins and Buffers, the property window shows the available properties directly. Once changes are made, click **OK** to close the **Property** window.

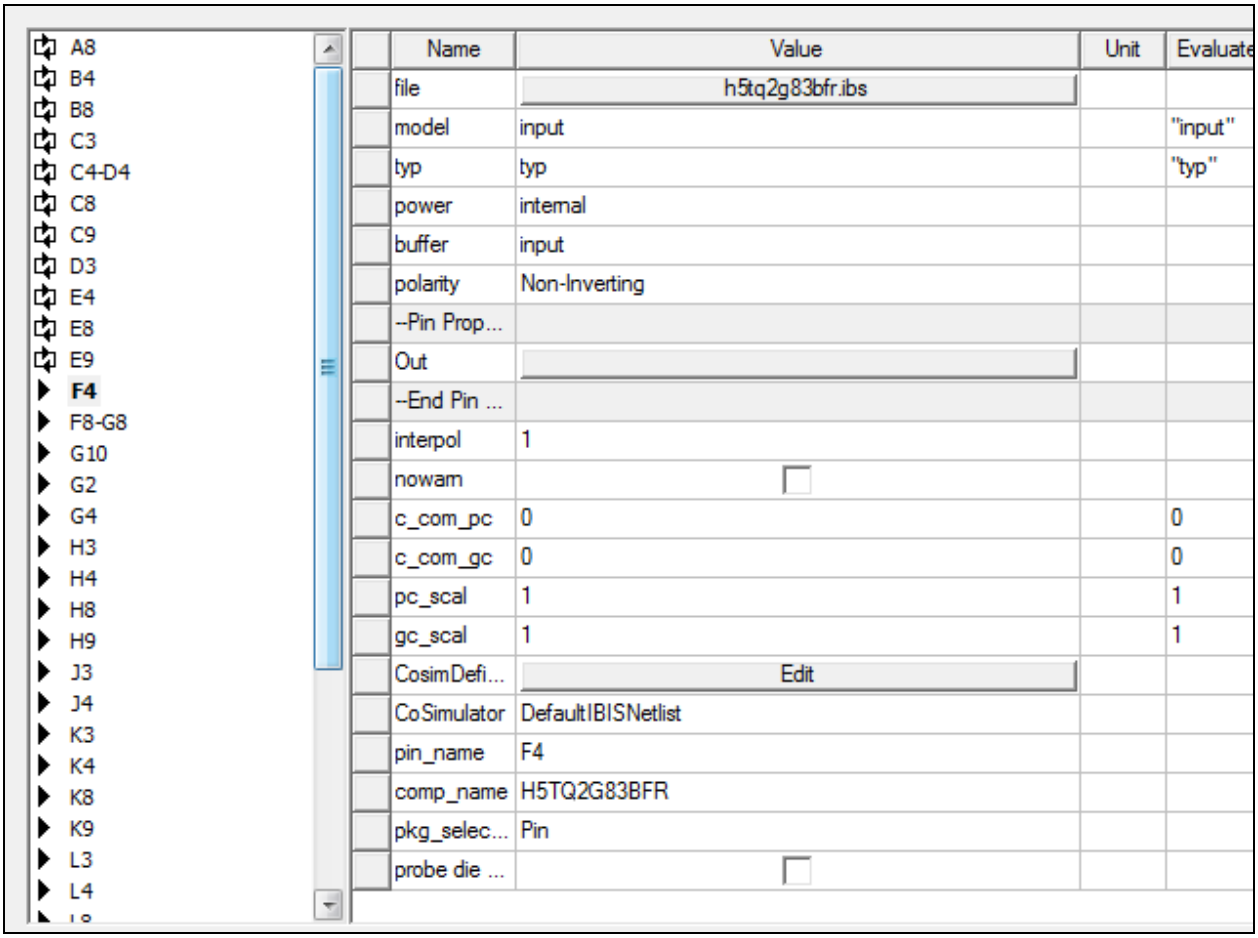
IBIS Property Locations

When the import is of an entire IBIS component, clicking **Properties** opens a Property window like the following:

	Name	Value	Unit	Eva...	Description
	file	m25p16_3v_it_ansys2.ibs			Name of IBIS file (required)
	CosimDefinition	Edit			
	Pin Info	Pin Info			
	CoSimulator	DefaultNetlist			
	Component External Power	☑			
	Package Model	M25P16_MW_S08W_ansys...			
	Status	Active			

From the **Properties** window for a single component, click **Pin Info** to bring up a window listing all the pins and the pin 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 area on the left lists all the pins in the component, with icons representing the buffer type. When you select one or more pins, the properties that apply to the selected pins are listed in the area on the right. The **Properties** window lists differential pin pairs (e.g., C4-D4), but the symbol shows separate terminals for both pins of a differential pair.

For a single pin or buffer model, all properties are on one page:

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	Eva...	Description	Ca
	file	m25p16_3...			Name of IBIS file (required)	
	model	SELECT		"SE...	Name of model (required)	
	typ	typ		"typ"	Parameter range (typ, min, max, fast, slow)	
	power	internal			Power mode (on=internal, off=external)	
	buffer	input			Buffer type (input)	
	polarity	Non-Invert...			Polarity (Non-Inverting, Inverting, No Polarity, ...	
	--Pin Properties					
	Out					
	--End Pin Properties					
	interpol	1			Interpolation (1=linear, 2=spline)	
	nowarn	☐			Disable parser warnings	
	c_com_pc	0		0	Fraction of capacitance on power clamp	
	c_com_gc	0		0	Fraction of capacitance on ground clamp	
	pc_scal	1		1	Scale factor for power clamp current drive	
	gc_scal	1		1	Scale factor for ground clamp current drive	
	CosimDefinition	Edit				
	pin_name	7				
	comp_name	M25P16-V...				
	pkg_selector	Package				
	probe die side	☐				
	CoSimulator	DefaultIBI...				
	Status	Active				

Basic Buffer Properties

The basic IBIS component properties are displayed for all imported IBIS buffers.

- **File**: The name of the IBIS file.
- **Typ**: Assumed range of corner values for IBIS response parameters (See TYP Parameter Effects in the IBIS File Import Technical Notes)
- **Buffer**: Displays the buffer type corresponding to the model:
- Input
- Output
- Tristate
- Input/Output
- Open Drain, Open Sink, Open Source
- I/O Open Drain, I/O Open Sink, I/O Open Source
- Input ECL

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

- Output ECL
- Tristate ECL
- Input/Output ECL
- Terminator
- Series, Series Switch
- **Model**: Name of an IBIS model to associate with this component (pin or model). When the IBIS file specifies a Model Selector, the Model property lists the models in the selector so the user may select one. The list of available properties depends on the type of the model selected.

  > **Note:**
  >
  > A text array variable may be used for the **model** property. The array must contain model names with the same buffer type.
  >
  > See: Adding a Project Variable Array.

- **Polarity**: Non-inverting, Inverting, No Polarity, or Not Set, as specified in the IBIS file for each model.
- **Waveform Properties** The HSPICE-compatible Waveform properties control the details of the waveform processing and other buffer parameters. All buffer imports display the subset of these properties applicable to the buffer type.

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

interpol	1			Interpolation (1=linear, 2=spline)
nowarn				Disable parser warnings
c_com_pc	0		0	Fraction of capacitance on powe
c_com_gc	0		0	Fraction of capacitance on groun
c_com_pu	0		0	Fraction of capacitance on pullup
c_com_pd	0		0	Fraction of capacitance on pulldo
pc_scal	1		1	Scale factor for power clamp cur
gc_scal	1		1	Scale factor for ground clamp cur
pu_scal	1		1	Scale factor for pullup current driv
pd_scal	1		1	Scale factor for pulldown current
xv_pu				Node name for pu_scal
xv_pd				Node name for pd_scal
ramp_fwf	2			Falling waveform (0=ramp, 1=sing
ramp_rwf	2			Rising waveform (0=ramp, 1=singl
fwf_tune	0.1		0.1	Falling waveform adjustment
rwf_tune	0.1		0.1	Rising waveform adjustment
rwf_scal	1		1	Falling waveform scale factor
fwf_scal	1		1	Rising waveform scale factor
spu_scal	1		1	Additional scale factor for pullup c
spd_scal	1		1	Additional scale factor for pulldow
rm_dly_rwf	0	s	0s	Trim leading edge of rising wavefo
rm_dly_fwf	0	s	0s	Trim leading edge of falling wavef
rm_tail_rwf	0	s	0s	Trim trailing edge of rising wavefo
rm_tail_fwf	0	s	0s	Trim trailing edge of fallling wavef

- See Notes on IBIS Waveform Properties in the IBIS File Import Technical Notes for details on these properties.

Component and Pin Import Properties

These properties appear for single component and pin imports, not for buffer model imports.

- **pin_name** — The pin name on the IBIS file.
- **comp_name** — The name of the component on the IBIS file.
- **DieSideNodePrefix** — Allows users to add a keyword to the **probe die side node** name to help group nodes. The node representing the die side of the package is automatically generated. For a single IBIS pin component, the node is named **DS_***diesidenodeprefix*_*pinname_signalname_ID*, where:
  - *diesidenodeprefix* is the value entered in the **DieSideNodePrefix** property.
  - *pinname* is the **[Pin]** name on the imported pin's entry in the IBIS [Pin] table.

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

- *signalname* is the **signal_name** on the imported pin's entry in the IBIS [Pin] table.
- *ID* is the unique instance number of the IBIS device on the schematic (e.g., 130).

For example: `DS_U24_C2_DQ0_130`For differential pin pairs, the names of the two nodes are formatted like a single IBIS pin component with the addition of **_pos** and **_neg** suffixes. For example: `DS_U24_C3_DQS_t_131_pos` `DS_U24_C3_DQS_t_131_neg`In this case, *pinname* and *signalname* come on the first (non-inverting) pin in the IBIS [Diff Pin] table entry for the imported differential pin pair.

- **probe die side node** — The die side node waveforms are available in simulation output. If voltage probes are present elsewhere in the schematic, check the **probe die side node** check box to have the die side nodal voltage(s) appear as plottable quantities in the reporter, similar to if a probe had been placed on the die side node(s) for this IBIS device.
- **use_series_model** — For components with differential pins, the property list includes a **use_series_model** check box. Check this box to have Nexxim automatically instantiate any Series model defined between the differential pins in a Series Pin Mapping table in the IBIS file. Click the **model** property to manually select the Series model.

Pin Properties for Input Buffers

For Input buffers, the **Out** Pin property sets the connection for the (pseudo-digital) output pin that responds to inputs. Click the **Out Value** field button to open a window. Select from the list of nets to connect to the **Out** pin.

Pin Properties for Output Buffers

The **logic_in** Pin property sets the connection for the incoming data source. Click the **logic_in Value** field button to open a window. Select **Internal Source** to set up an internal data source (See *Eye Source Parameters*.) Click **Select Net** to display a list of nets to connect to the **logic_in** pin.

Eye Source Parameters When the IBIS buffer **logic_in** Pin property is set to **Internal Source**, the Property list includes selected Eye Source parameters to set up the internal data source.

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

--Eye Source Parameters			
trise	5e-010	s	5e-010s
tfall	5e-010	s	5e-010s
phase_delay	0	s	0s
UIorBPS	UnitInterval		
UIorBPSValue	1e-009	s	1e-009s
DCDFractionorTime	Fraction		
dcd	0		0
bxrj			
bxpj			
bxuj			
bxcj			
Disable_Tx_Jitter	☐		
repeat_count	0		0
step_resp_num_ui	100		100
do_encoding	0		0
hold_last_bit	☐		
BitPattern			
--End Eye Source Parameters			

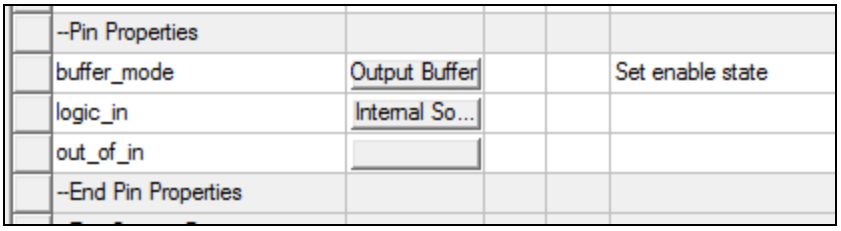
With the following exceptions, these parameters are documented in the topic Add a QuickEye Source to a Schematic.

Note:

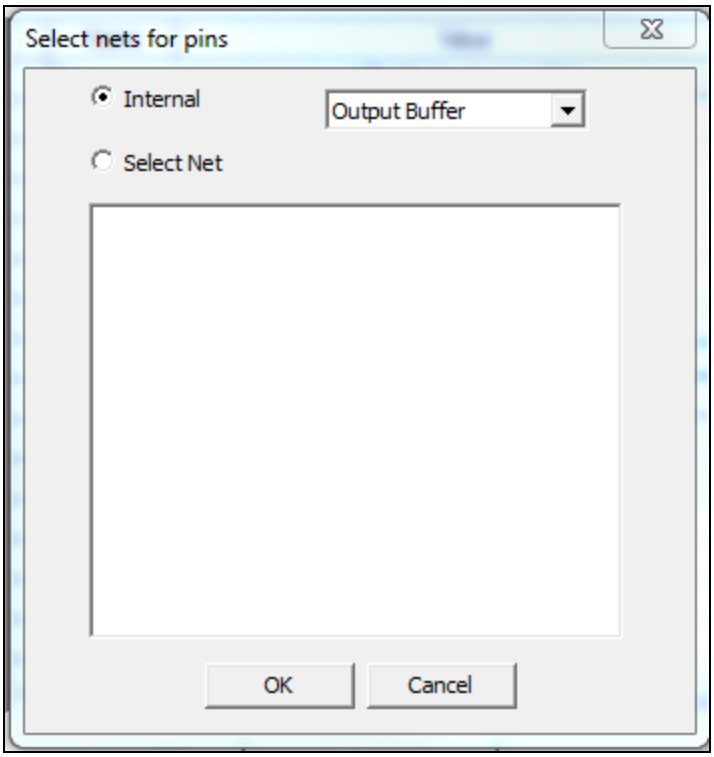
1. The **resistance** parameter on the Eye Source is not available with IBIS sources.
2. The **vlow** parameter on the Eye Source is not available with IBIS sources. IBIS sources must use 0V as the logic low voltage.
3. The **vhigh** parameter on the Eye Source is not available with IBIS sources. IBIS sources must use 1V as the logic high voltage.
4. The **FFE** parameter (Feed-Forward Equalization) on the Eye Source is not available with IBIS sources.
5. For IBIS, the **phase_delay** is a pure delay added before the bit pattern starts running.
6. The **Disable Tx Jitter** check box is available only on IBIS buffers. When this check box is **On** (checked), transmit jitter calculations are disabled. Transmit jitter settings in the properties list are not changed.

Pin Properties for Input-Output Buffers

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



The **buffer_mode** property sets the connection and condition for the Enable pin on the component (valid only for Input/Output and Tristate buffers). Click the **buffer_mode** property **Value** field button to open a window.



Select **Internal** to specify internal connections for the Enable pin. When **Internal** is checked, select buffer mode. The internal options depend on the **buffer_type** basic property:

- For **Input_Outputbuffer_types**, select Output Buffer or Input Buffer (default Output Buffer).
- For **Tristatebuffer_types**, select Output Buffer or High Impedance Buffer (default Output Buffer).

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

Alternatively, click **Select Net**. The window displays a list of nets in the circuit to connect to the Enable pin. The text at the upper-right of the window shows the setting of the Enable specification (Active-High, Active-Low) as read on the IBIS model in the file.

Click **OK** to close the window.

The **logic_in** Pin property sets the connection for the incoming data source. Click the **logic_in Value** field button to open a window. Select **Internal Source** to set up an internal data source (Refer to *Eye Source Parameters*.) Click **Select Net** to display a list of nets to connect to the **logic_in** pin.

For Input_Output buffers, the **out_of_in** Pin property sets the connection for the (pseudo-digital) output pin that responds to inputs. Click the **out_of_in Value** field button to open a window. Select from the list of nets to connect to the **out_of_in** pin.

Package Support Properties

Package Model This property is available only with a single component import that references a Package Model. The name of the [Package Model] definition to use for instantiating the RLC parasitics. With a [Package Model], Nexxim is able to instantiate the coupled RLC parasitics between the die side and the board side of each pin. When a [Package Model] is used, the **pkg_selector** property is not displayed for the individual pins.

pkg_selector The **pkg_selector** parameter is used to select the appropriate packaging model for pins not handled by a [Package Model] definition. When **pkg_selector** is set to **Pin** (the default), the pin-specific package model is used if it exists; otherwise, the default package model for that component is used. When **pkg_selector** is set to **Package**, the default [Package] model for that component is used, whether a [Pin] package model exists. When **pkg_selector** is set to **None**, no package model is used, and the device behavior is identical to one created with a Buffer Import of the same model.

Power Selection Properties

Component External Power This property is available only with a single component import that references a Package Model. When **Component External Power** is **On** (checked), Nexxim expects power via the terminal pins on the component schematic symbol, and the **power** property is ignored for the individual pins. Nexxim connects power buses as described in the component [Pin Mapping] entry.

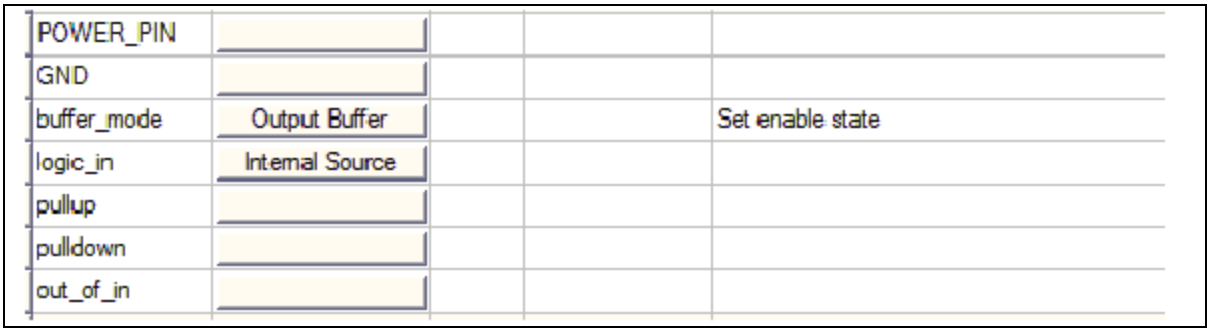
> **Note:**
>
> If you subsequently deactivate (uncheck) **Component External Power**, the power to each pin must be specified with its **power** property (on the **Pin Info** window), and any POWER or GND pins that are imported become No-Connects.

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 subsequently deactivate (uncheck) **Component External Power**, the power to each pin must be specified with its **power** property (on the **Pin Info** window), and any POWER or GND pins that are imported become No-Connects..

Power Connections for power nodes, internal or external. This parameter is enabled when a single component with a package model has the Component External Power deactivated, and for all imported single pins and buffer models.

When the **power** property is set to **external**, the Pin Properties show the power pins:



Click the power, ground, pull-up, and pulldown **Value** field buttons to open the window, then select on the available nets.

> **Note:**
>
> Use page ports to name the outputs of sources in the circuit as nets.

For more information, see *Power Nodes* in the "IBIS File Import Technical Notes" below

AMI Specific Properties

When the IBIS Pin Import selection references an Algorithmic Model (AMI), the properties list contains AMI-specific properties including AMI file references and AMI parameters that are made user-visible in the AMI file. Refer to AMI Analysis for details on importing AMI components.

IBIS File Import Technical Notes

This topic describes the conventions that should be followed to produce a correct simulation with the imported IBIS buffer elements.

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

Support for IBIS Specification

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform can parse IBIS® input files, but not all features of the IBIS specification are supported. Some IBIS buffers are available as built-in Nexxim components.

AEDT supports the following features of the IBIS specification:

- Import IBIS library files (.ibs).
- Import IBIS Electrical Board Description files (.ebd); see Importing IBIS EBD Files.
- Convert models for single-ended (not differential) driver and receiver pins to SPICE circuits.
- Extract IV curves on the following and translate into equivalent piece-wise linear devices for SPICE: Pull-up, Pull-down, power and ground clamp devices.
- Extract rise/fall-time information on the IBIS file in order to sequence the pull-up/down devices.
- Read in pin parasitics (RLC elements) and incorporate into models.
- Package models ([Package Model] coupled RLC matrices, .pkg files.)
- Transmitter and receiver analog models from Touchstone files (Note: BIRD158 is provisional. It is not yet approved into the IBIS standard and is subject to change. The Ts4 related parameters that are supported represent everything up to and including revision BIRD 158.7).

The following features of the IBIS specification are **not** supported:

- Series switch groups
- Submodels of type Bus_hold and Fall_back. Dynamic_clamp IS supported.
- Power/ground pulse tables

Power Nodes

All active IBIS components contains some combination of pullup, pulldown, power clamp, and ground clamp nodes. Power to these nodes can be applied at the component level using the [Pin Mapping], or at the relevant nodes of each pin, using the default power parameters.

Component-Level Power and Ground with Package Models

When you are importing a Single Component with a Package Model, the power and gnd pins that you select are exposed in the schematic to wire to the selected power or ground rails. Nexxim uses the [Pin Mapping] information (power and ground bus definitions) in the IBIS component definition along with the RLC parasitic data (from die side to board side) in the Package Model definition to connect the power and ground buses to power and ground nodes on the signal pin buffers. The **Component External Power** property is set to ON in the

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

component. When **Component External Power** is ON, the power parameters on individual pins is ignored.

The IBIS file may contain a [Pin Mapping] keyword defining buses that connect POWER and GND pins to the voltage supply nodes of any buffers connected to signal pins. When a [Package Model] is used, and the **Component External Power** is on, Nexxim uses the [Pin Mapping] data to connect the prescribed power buses between the die-side nodes of POWER and GND pins and the voltage supply nodes of buffers. Any imported pins that are NOT included in the Package Model definition use the uncoupled lumped RLC package defined in their [Pin] entry (or [Package] section if no [Pin] RLC data is specified).

Pin-Level Power and Ground

When you are importing either a Single Component without a Package Model, individual pins, or individual buffer models, power connections click the **power** parameter on the individual pins to control the connections. The **power** property also controls the power connections when a single component with a [Package Model] has been imported but the **Component External Power** switch is deactivated; changing the imported POWER and GND pins to No-Connects (NC).

By default, the **power** parameter is set to **internal**, so the component automatically drives the nodes using internal voltage sources set to the voltages specified in the IBIS model. When **power** is set to **internal**, users should NOT connect any of these nodes to voltage sources or to ground. Connecting external voltage sources or grounds when **power** is set to **internal** generates warning messages about parallel supplies, and may cause problems with simulation.

To use external sources and grounds, set the **power** parameter to **external**. With this setting, any pull-up, pulldown, power clamp, and ground clamp nodes appear as pin parameters with lists for selecting the external voltage nodes to connect (using page ports).

Package RLC Parasitics

Each pin on the component has two nodes, a die-side node and a board-side node (See the **probe_die_side** parameter discussion in Edit IBIS Component Properties). You can probe both nodes, but you can connect only to the board-side node. Between the die side and the board side, the package introduces RLC parasitics. Accurate modeling of these parasitics is required for signal and power integrity analyses.

By default, the RLC data is specified in the **[Package]** keyword in the IBIS file. The **[Package]** entry defines a range of values for the default RLC of the component pins. The IBIS file **[Pin]** keyword can contain values for the RLC parasitics on a pin-by-pin basis. If the **[Pin]** entry for the component includes individual RLC values for a given pin, they override the defaults in **[Package]**, for that pin. The **pkg_selector** property defaults to **Pin**. With this setting, Nexxim searches the IBIS component definition for a **[Pin]** keyword containing RLC data for the selected pin, and applies that data if the **[Pin]** entry is found. If no **[Pin]** entry is found, Nexxim uses the default **[Package]** data. Also select **Package** to have Nexxim skip the search for a **[Pin]** entry.

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 IBIS file contains a [Package Model] keyword and you select **Use [Package Model]** at import time, the model RLC values override the **[Pin]** and **[Package]** values for the imported pins that are included in the Package Model Definition. The **pkg_selector** property is not displayed for the imported pins that have a **[Package Model]** definition. Any imported pins that are NOT included in the Package Model definition are handled as though their **pkg_selector** is set to **[Pin]**).

Support for Power Aware IBIS Output Buffer Models

Traditional IBIS models supply tables of currents delivered to the output by the pullup, pulldown, power clamp and ground clamp structures. The current delivered at the output is the sum of those four contributions. With traditional IBIS models, the current drawn on the power rails is whatever the four structures draw to deliver to the output. Power Aware IBIS output buffer models contain additional information about the actual current drawn on the supply rails. They also contain information that allows the simulator to more accurately account for the effects of variations in the supply rail voltages.

Composite Current

A **[Composite Current]** section in the IBIS file describes the shape of the rising or falling edge waveforms in the total current through the power reference terminal of the buffer. The composite current (Icc), the total current to be drawn through the power pin, is the sum of the current at the output (Iout) and the current internal to the device that goes to ground without reaching the output (Ibypass):

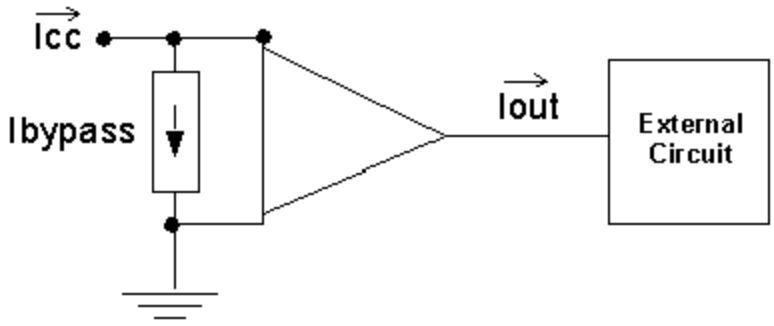
$$Icc = Iout + Ibypass$$

The IBIS file contains **[Rising Waveform]** and **[Falling Waveform]** tables of times and output voltages for the waveforms. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform calculates the output current Iout based on the output voltage and the terminating load.

When the IBIS model file for an Output buffer contains a **[Composite Current]** section after a **[Rising Waveform]** or **[Falling Waveform]** table, Nexxim reads in the table of times and currents. The times in the composite current table are identical to the times in the corresponding waveform table. Nexxim calculates Ibypass at each time point by subtracting the output current on the composite current.

During simulation, Nexxim adds a current source to generate Ibypass between the power and ground pins of the power-aware Output buffer.

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 direction for Iout in the diagram has been reversed on the convention used in the IBIS specification, in order to simplify the formulas and discussion.

The current drawn on the power pin equals Icc, the current specified in the **[Composite Current]** table. The output current, Iout, is unchanged.

The composite current feature allows you to simulate with a non-ideal power supply and get accurate values for the power consumption. Power usage during events such as simultaneous switching operations (SSO) can be simulated accurately.

Gate Modulation Effect

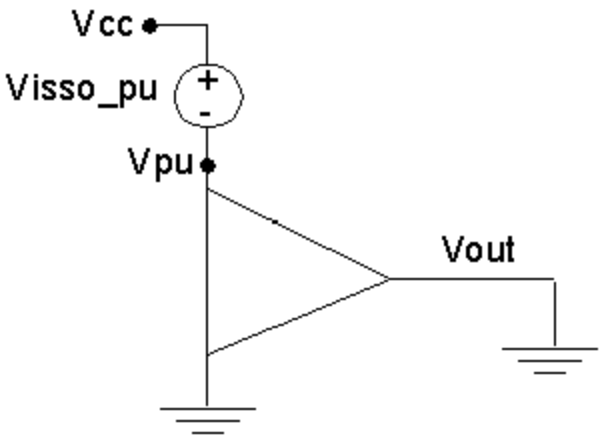
The gate modulation effect is the change in output current Iout due to fluctuations in the gate voltage.

The **[ISSO PU]** and **[ISSO PD]** tables allow the simulation to account for variations in the gate current due to bouncing on the supply voltage.

Gate Modulation at the Pullup Node

The **[Pullup]** section provides tables of output current Iout for values of the voltage drop (Vpu - Vout). Here is a reference diagram.

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 absence of gate modulation:

$$Iout = Pullup(Vpu - Vout)$$

Vpu need not be equal to ideal voltage Vcc. Any voltage drop at the power pins affects the voltage difference between the power pin and the output pin, affects the lookup in the I-V table for the model.

The **[ISSO PU]** table gives the effective current of the pullup structure as a function of the voltage on the pullup reference node (the power node). When Nexxim finds an **[ISSO PU]** section after a **[Pullup]** entry, the calculation for Iout proceeds as follows.

1. Nexxim calculates the voltage drop between Vpu and Vout and uses this value as before to look up Iout before the adjustment for gate modulation.
2. Nexxim calculates the voltage difference between the ideal Vcc and the actual pullup voltage Vpu. This voltage difference is called Visso_pu in the diagram.
3. Nexxim uses the value of Visso_pu to access the table in the **[ISSO PU]** section. The table gives the output current for that amount of voltage modulation, Iisso_pu.
4. Nexxim accesses the **[ISSO PU]** table entry for Visso_pu=0V. The value of Iout at Visso_pu=0 is the nominal current, Inom_pu.
5. Nexxim calculates a scale factor Kpu as the ratio of the tabulated output current to the nominal current:

$$Kpu = \frac{Iissopu}{Inompu}$$

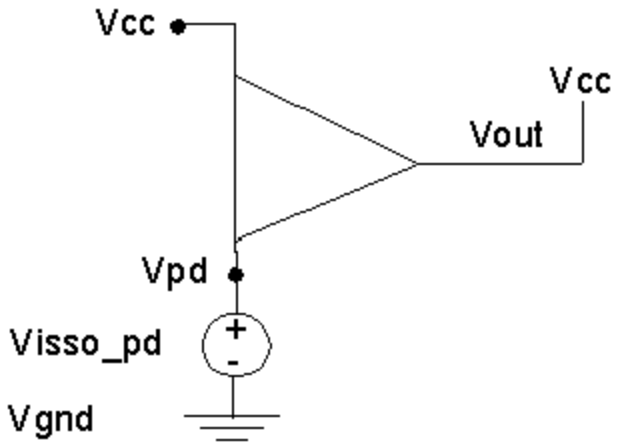
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. The final output current is the ideal output current on the **[Pullup]** table, scaled by the Kpu factor for the voltage modulation on the **[ISSO PU]** table:

$$Iout = Pullup(Vpu - Vout) \times Kpu(Vcc - Vpu)$$

Gate Modulation at the drop-down menu Node

The **[Pulldown]** section provides tables of output current Iout for values of the voltage drop on the output voltage to the drop-down menu reference voltage (Vout - Vpd). Here is a reference diagram.



From the absence of gate modulation:

$$Iout = Pulldown(Vout - Vpd)$$

The **[ISSO PD]** table gives the effective current of the drop-down menu structure as a function of the voltage from the drop-down menu reference node (the ground node).

When Nexxim finds a **[ISSO PD]** section after a **[Pulldown]** entry, the calculation for Iout proceeds as follows.

1. Nexxim calculates the voltage drop between Vout and Vpd and uses this value as before to look up Iout before the adjustment for gate modulation.
2. Nexxim calculates the voltage difference between the actual pulldown voltage Vpd and the ideal ground reference Vgnd. This voltage difference is called Visso_pd in the diagram.

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

3. Nexxim uses the value of Visso_pd to access the table in the **[ISSO PD]** section. The table gives the output current for that amount of voltage modulation, Iisso_pd.
4. Nexxim accesses the **[ISSO PD]** table entry for Visso_pd=0V. The value of Iout at Visso_pd=0 is the nominal current, Inom_pd.
5. Nexxim calculates a scale factor Kpd as the ratio of the tabulated output current to the nominal current:

$$Kpd = \frac{Iissopd}{Inompd}$$

6. The final output current is the ideal output current on the **[Pulldown]** table, scaled by the Kpd factor for the voltage modulation on the **[ISSO PD]** table:

$$Iout = Pulldown(Vout - Vpd) \times Kpd(Vpd - Vgnd)$$

Digital Voltage Levels

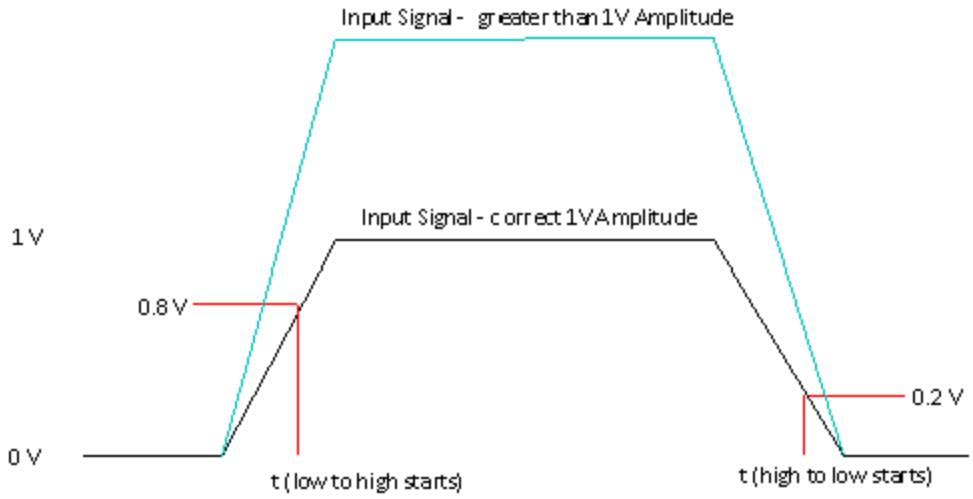
All digital nodes on IBIS components expect a voltage range of 0V to 1V.

When the internal buffer_mode settings and the internal eye source are used, the proper voltage levels are already assured. However, if you are connecting external nets to the "enable" or "logic_in" nodes instead, it is important to use the proper voltage range on the input signals.

Input Node Thresholds

Digital input nodes for drivers (output, I/O, etc.) use fixed thresholds of 0.2V and 0.8V to trigger changes in state from high to low and low to high, respectively. (See 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.



An external voltage supply may be connected to these nodes, but must supply the expected values: 0V for low or off and 1V for high or on. Because the transition trigger thresholds are fixed, duty cycle distortion (DCD) occurs if a high amplitude other than 1Vis used (blue trace in diagram). The amount of DCD is negligible if the transitions themselves are very fast relative to the overall pulse width. Using a high amplitude of less than 0.8V results in no output transitions occurring.

For initial transient values, and for DC analysis, a single fixed input threshold of 0.5V is used. If the input signal is above 0.5V, the output is placed in the high state. Otherwise, the output is placed in the low state.

These descriptions of the relationships between input voltage and output state for the Input buffer nodes assume the **Polarity** parameter in the IBIS model is set to **Non-Inverting**. The relationships are appropriately inverted if the IBIS buffer model has **Polarity** defined as **Inverting**.

Output Node Thresholds

The digital output nodes of receivers provide a waveform that switches between 0V and 1V. These transitions depend on the input signal, the values of **Vinl**, **Vinh**, and **Polarity** set in the IBIS model file, and the following criteria:

- If the output is in the low state (0V) and the input signal becomes greater than **Vinh**, the output transitions to high (1V).
- If the output state is high (1V) and the input signal becomes less than **Vinl**, the output transitions to low (0V).

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

- For initial transient values, and for DC analysis, the buffers use a single fixed threshold equal to the average of **Vinl** and **Vinh**, [(**Vinh** + **Vinl**)/2]. If the input signal is above this threshold, the output is placed in the high state (1V). Otherwise, the output is placed in the low state (0V).

These descriptions of the relationships between input voltage and output state for the Output buffer nodes assume the **Polarity** parameter in the IBIS model is set to **Non-Inverting**. The relationships are appropriately inverted if the IBIS buffer model has **Polarity** defined as **Inverting**.

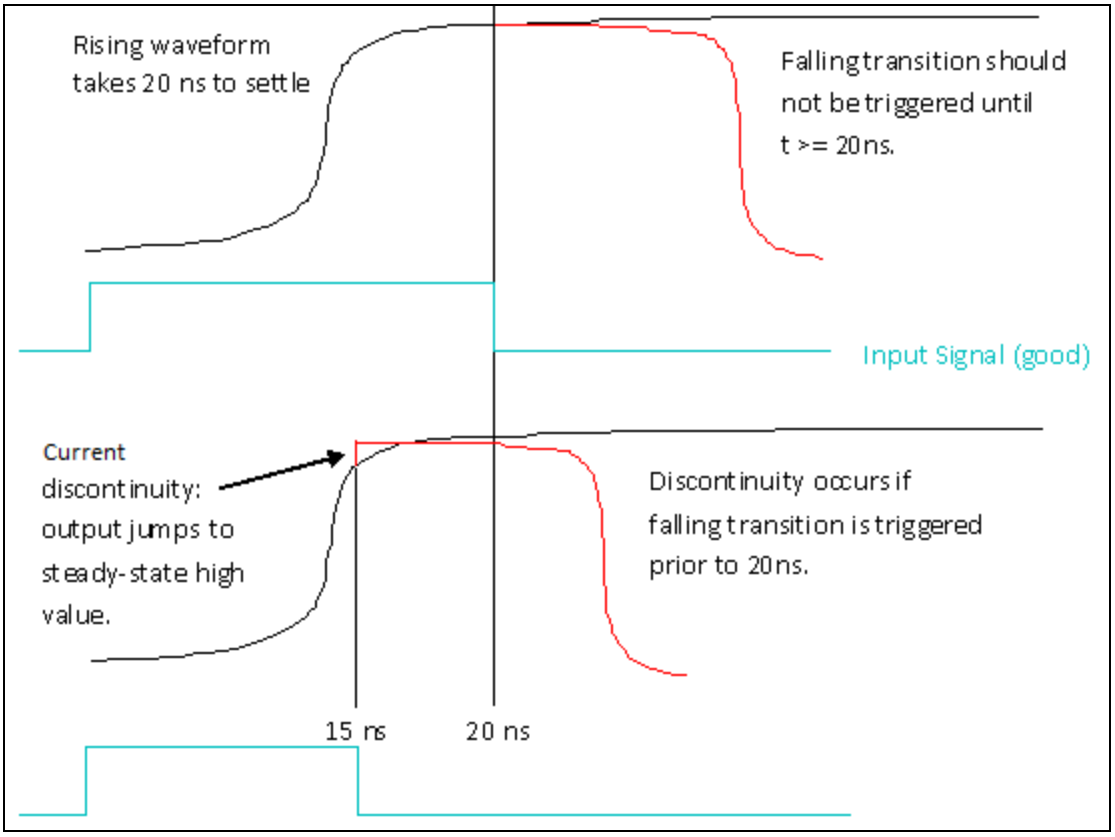
Enable Node Thresholds

A single fixed threshold of 0.5V is used for Enable nodes. If the Enable pin is higher than 0.5V, the enable state is considered high. Other wise, the enable state is considered low. The behavior of the component then depends on whether the IBIS buffer model has its **Enable** parameter set to **Active-High** or **Active-Low** in the IBIS file.

Transition Settling Times

Be careful to ensure the frequency of the input signal does not exceed the capabilities of the IBIS model. The frequency of the input signal must consider the time required for the output to settle. Whenever a transition from high to low or from low to high is triggered by the input signal, the output jumps to the steady-state current value from which the next transition starts. However, the actual waveform requires a settling time period to reach the steady-state high or low voltage. If the input then changes state too soon, the next transition occurs before the initial transition has settled to its steady-state value. This can lead to non-physical voltage slew rates and other unexpected 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.



For example, if the IBIS model takes 20ns to complete a rising transition, the input signal should stay in the high state (1V) for at least 20ns after a low-to-high-transition. If instead the input goes back to low after only 15ns, the behavior at the high-to-low transition creates a non-physical slope change in the output signal. Since the initial rising transition has not completed at 15ns, the output has not settled at the steady state from which the falling transition is to occur. The too-early falling transition causes the output current to jump discontinuously to the high value.

Notes on IBIS Waveform Properties

The IBIS buffer component Waveform properties allow for fine-tuning the power scaling and waveform rise and fall. The following table summarizes the property settings. Detailed notes on some of these properties follow the table.

IBIS Buffer Waveform Properties

Property	Description	Units	Default
INTERPOL	Interpolation: 1=linear, 2=spline	None	1
NOWARN	When present, turns off parser warnings	None	Warnings enabled
XV_PU	Name of node for pullup scaling factor output	None	None

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

XV_PD	Name of node for pulldown scaling factor output	None	None
RAMP_FWF	Falling waveform ramp specification: 0=ramp only, 1=single waveform in IBIS file, 2=double waveform in IBIS file	None	2
RAMP_RWF	Rising waveform ramp specification: 0=ramp only, 1=single waveform in IBIS file, 2=double waveform in IBIS file	None	2
FWF_TUNE	Falling waveform adjustment. Used only when RAMP_FWF=0 or 1. Waveform assumes device transitions from on to off in the fraction of the total transition time given by FWF_TUNE.	None	0.1
RWF_TUNE	Rising waveform adjustment. Used only when RAMP_RWF=0 or 1. Waveform assumes device transitions from off to on in the fraction of the total transition time given by RWF_TUNE.	None	0.1
C_COM_PC	Fraction of capacitance on power clamp vs. GC, PU, and PD, (value between 0 and 1). Capacitance fractions must sum to 1.0.	None	0
C_COM_GC	Fraction of capacitance on ground clamp vs. PC, PU, and PD (value between 0 and 1). Capacitance fractions must sum to 1.0.	None	1
C_COM_PU	Fraction of capacitance on pull-up vs. GC, PC, and PD (value between 0 and 1). Capacitance fractions must sum to 1.0.	None	0
C_COM_PD	Fraction of capacitance on pulldown vs. PC, GC, and PU (value between 0 and 1). Capacitance fractions must sum to 1.0.	None	1
PC_SCAL	Scale factor for power clamp current drive strength applied to IBIS table values	None	1.0
GC_SCAL	Scale factor for ground clamp current drive strength applied to IBIS table values	None	1.0
PU_SCAL	Scale factor for pull-up current drive strength applied to IBIS table values	None	1.0
PD_SCAL	Scale factor for pulldown current drive strength applied to IBIS table values	None	1.0
FWF_SCAL	Scale factor applied to falling ramp transition time	None	1.0
RWF_SCAL	Scale factor applied to rising ramp transition time	None	1.0
SPD_SCAL	Scale factor for pulldown current drive strength applied to	None	1.0

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

	IBIS table values Used only when POWER=OFF and the voltage difference between the pull-up and pulldown nodes is not equal to the [Voltage Range] value in the IBIS file.		
SPU_SCAL	Scale factor for pull-up current drive strength applied to IBIS table values Used only when POWER=OFF and the voltage difference between the pull-up and pulldown nodes is not equal to the [Voltage Range] value in the IBIS file.	None	1.0
RM_DLY_RWF	Amount of time to trim off the front of the rising waveform, relative to time=0. Negative value invokes automatic removal of common delays on the fronts of rising and falling waveforms [see Automatic Delay Removal].	>=0: Sec <0: None	0.0
RM_DLY_FWF	Amount of time to trim off the front of the falling waveform, relative to time=0 Negative value invokes automatic removal of common delays on the fronts of rising and falling waveforms [see Automatic Delay Removal].	>=0: Sec <0: None	0.0
RM_TAIL_RWF	Amount of time to trim off the end of the rising waveform, relative to the time of the last point in the waveform	Sec	0.0
RM_TAIL_FWF	Amount of time to trim off the end of the falling waveform, relative to the time of the last point in the waveform	Sec	0.0

[1] If the values of RM_DLY_RWF and RM_DLY_FWF are not equal, Nexxim generates a warning, since the unequal values alter the duty cycle of the output waveform relative to what is specified by the IBIS file itself. The simulation runs.

[2] If the value of RM_DLY_RWF, RM_DLY_FWF, RM_TAIL_RWF, or RM_TAIL_FWF is such that more than 10 percent of the original voltage transition is trimmed off, Nexxim generates a warning, but the simulation runs.

[3] If the values of RM_DLY_RWF and RM_TAIL_RWF are such that fewer than three (3) data points are left in the resulting waveform, Nexxim ignores the settings and issues a warning. With fewer than three data points, ramp transitions may give better results.

[4] If the values of RM_DLY_FWF and RM_TAIL_FWF are such that fewer than three (3) data points are left in the resulting waveform, Nexxim ignores the settings and issues a warning. With fewer than three data points, ramp transitions may give better results.

[5] If the sum of RM_DLY_RWF and RM_TAIL_RWF is such the entire rising waveform is removed, Nexxim halts with an error.

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 sum of RM_DLY_FWF and RM_TAIL_FWF is such the entire falling waveform is removed, Nexxim halts with an error.

Automatic delay removal. Some IBIS models represent the throughput delay of the device by providing model waveforms with long initial plateaus, after which the device is switched at a much shorter period corresponding to the actual slew rate. However, the slew rate observed in Nexxim can be too high in such cases because the initial delay is not recognized as such, and the device appears to transition directly from steady-state low to steady-state high (and vice versa) when it is switched at the shorter period. Using RM_DLY_RWF and RM_DLY_FWF to manually trim the initial delay can be cumbersome when the model's waveforms have widely differing initial delays according to corner case (typ, min, max, fast, slow). The overall timing of the simulation is also shifted by trimming the waveforms.

Setting either RM_DLY_RWF or RM_DLY_FWF to a negative value (<0) invokes automatic delay removal for both rising and falling waveforms. The initial delay plateau is computed for each waveform in a given device's IBIS model. The common delay is the minimum of the values computed for all waveforms for the selected corner case (typ, min, max, fast, slow). The common delay time is removed on the beginning of each waveform. Then, an ideal delay element is added into the simulation flow to adjust for the time trimmed off the front of the waveforms.

TYP Parameter Effects

The **TYP** Parameter sets the allowable range of values affecting multiple IBIS buffer parameters:

typ = midrange of all values

min = minimum of all values

max = maximum of all values

fast = maxima and minima selected to produce the fastest transitions.

slow = maxima and minima selected to produce the slowest transitions.

The following table lists the effects on IBIS buffer parameters when the **TYP** parameter is set to **Fast** or **Slow**. In general, the **Fast** setting sets maxima and minima to provide the fastest transitions, while the **Slow** setting seeks to provide the slowest transitions.

Effects of TYP = Fast or TYP = Slow

IBIS Buffer Parameter	TYP Setting	
	Fast	**Slow**
C_comp	Min	Max
Cac	Min	Max
GND Clamp Reference	Min	Max
Pulldown Reference	Min	Max

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

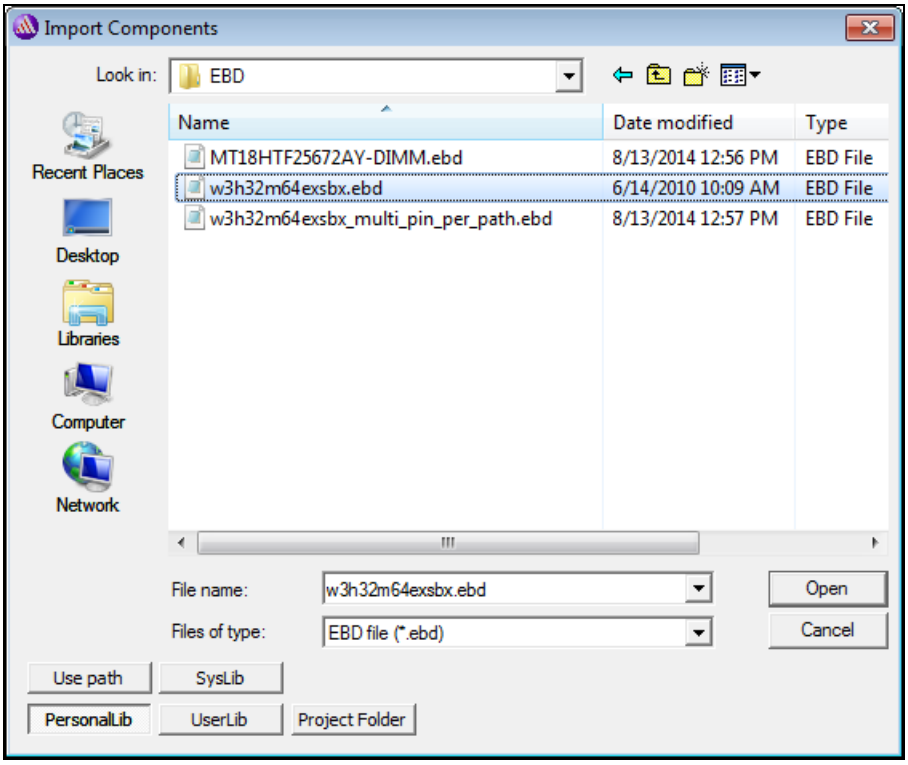
IBIS Buffer Parameter	TYP Setting	
	Fast	Slow
R Series	Min	Max
L Series	Min	Max
RI Series	Min	Max
C Series	Min	Max
Lc Series	Min	Max
Rc Series	Min	Max
Temperature Range	Max	Min
Voltage Range	Max	Min
Pullup Reference	Max	Min
POWER Clamp Reference	Max	Min
Rgnd	Max	Min
Rpower	Max	Min
Pulldown	Max	Min
Pullup	Max	Min
GND Clamp	Max	Min
POWER Clamp	Max	Min
Ramp	Max	Min
Rising Waveform	Max	Min
Falling Waveform	Max	Min
Series Current	Max	Min
Series MOSFET	Max	Min
V_fixture	Max	Min

Importing IBIS EBD Files

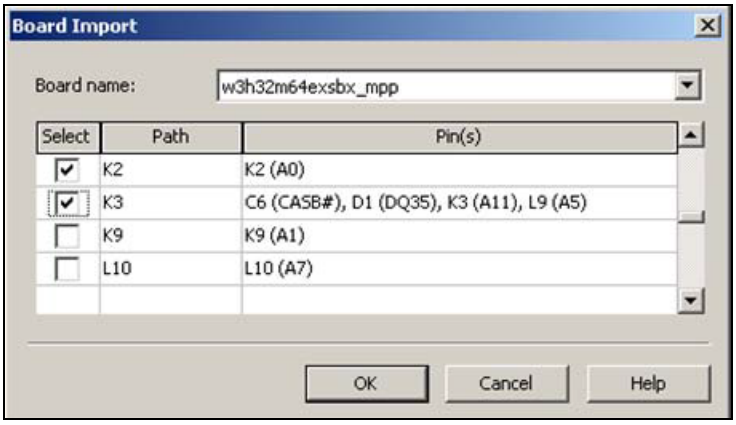
The Ansys Electronics Desktop (AEDT)™ electronics systems design platform IBIS support allows you to import.ebd files that contain board descriptions in the form of paths. When EDB file paths are imported, a corresponding Nexxim component is created. This option is only available for Nexxim designs. To import an IBIS EBD model:

1. View the **Component Libraries**. Open the **Symbols** area. Click **Import Models**.
2. From the area, select the **EBD** icon to open the **Import Components** window.

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



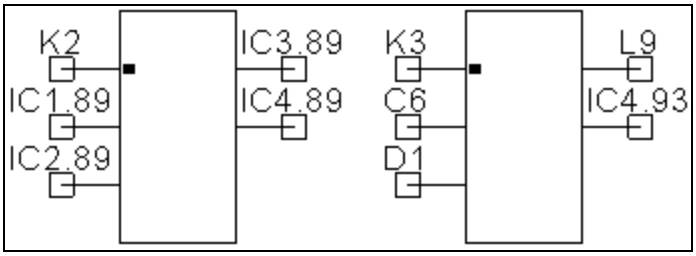
Select an .ebd file and click **Open** to open the following window. From the window, select boards and paths to import the file:



The **Board name** drop-down menu displays all the boards contained in the .ebd file; when a board is selected, the display grid updates to show the paths and pins contained on that board. The display grid also lists the various signal names for each pin in parenthesis. After selecting the selected paths and clicking **OK**, components are generated , then displayed 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.

Components tab and in the Definitions/Components area of the **Project Tree**. When placed on the schematic, the generated components resemble the following:



IBIS I/O Wizard

The I/O Wizard provides an easy interface to set up drivers and receivers for a single schematic or for a group of schematic components. The drivers and receivers the I/O Wizard uses are IBIS components that have been previously imported. (If you have not already loaded the IBIS models, you are given the option to do so by the I/O Wizard.). See Import IBIS Library File.

The I/O Wizard inspects the current schematic, indicating all unconnected pins, and allows you to create components for the selected connectivity configuration. Connectivity options include:

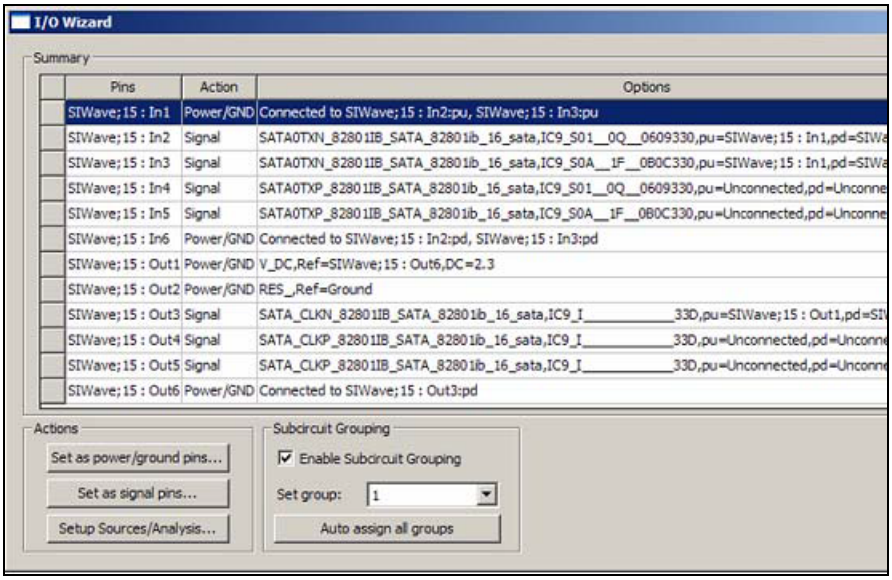
- Fully configured IBIS drivers and receivers
- Power and Ground pins
- Probes and designated points of circuit stimulation, such as Pseudo Random Bit Sources

After you set the initial configuration, the I/O Wizard can be rerun and any newly added or unconnected pins are indicated, allowing you to further modify the design.

IBIS I/O Wizard window

The I/O Wizard window provides a summary of the current Wizard configuration and allows you to perform various actions in order to set up individual pins. Click **Schematic** > **I/O Wizard** to open the Wizard window.

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



Upon initialization, the I/O Wizard indicates all the unconnected pins that exist in the current schematic. For example, in the preceding screenshot, the Wizard window after each pin of a 12-pin Ansys SIwave™ component has been configured.

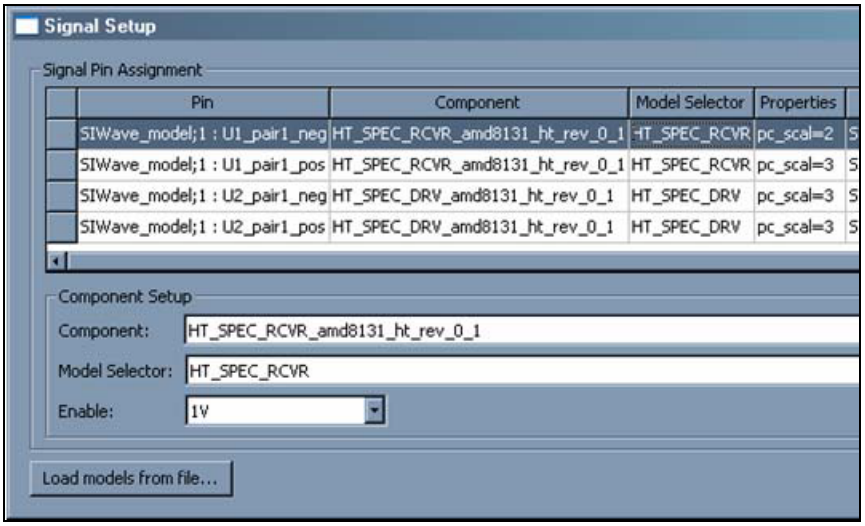
- The Wizard displays the current settings for each pin.
- Pins can either be left in the default unassigned state, assigned to a signal pin (driver or receiver), or assigned as a Power/Ground pin.
- Select one or more rows and click the various **Actions** buttons to associate selected pins with a given type. A configuration window appears that allows further customization.
- Power/Ground pins are initially assigned if the component's name contains "VSS", "GND", etc. Initial assignments can be overridden and customized.
- When **Enable Subcircuit Grouping** is selected, pins can be assigned to a group.
- **Auto assign all groups** attempts to group pins in a meaningful way — by grouping buffers with common pins into the same subcircuit group.

Click **OK** to close the window and implement any changes that have been configured.

IBIS I/O Wizard Signal Setup

To configure signal pin assignments using the I/O Wizard window, select the appropriate pins and click **Set as signal pins** ; this opens the **Signal Setup** window for the selected pins.

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



Fields in the **Signal Setup** window are editable. Also select multiple rows and click the **Component Setup** fields at bottom to make multiple changes to many pins simultaneously.

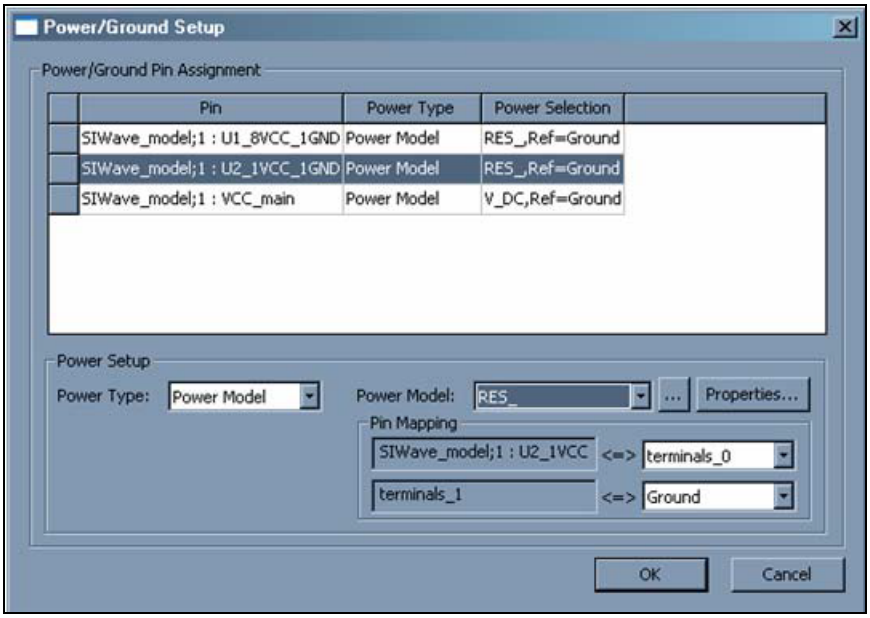
- The **Component** field lists components already imported, and represents either an IBIS pin or an IBIS model, depending on the component selected.
- The **ModelSelector** field lists the available models for this component (e.g., the **model_sel** property of the IBIS component).
- The **Pullup to pin** field lists non-signal pins (e.g., Power or Ground) to connect to the **Pull Up** pin for the attached buffer. The default is a Power/Ground pin. Alternately, select to select any one-port circuit in the project for the connection.
- The **Pulldownto pin** field follows the same rules as **Pullup to pin**, with regards to the **Pull Down** assignment for the attached buffer.
- For I/O Buffers, there is an **Enable** field that can either be 0V or 1V (depending on how the buffer is configured to operate). The **Enable** field is only visible for I/O buffers.
- Also load additional IBIS files by clicking **Load models from file** to open the Import IBIS File window.

Click **OK** to close the window and implement any changes that have been configured.

IBIS I/O Wizard Power/Ground Pin Setup

To configure Power and Ground settings, click **Set as power/ground pins** in the I/O Wizard window; this opens the **Power/Ground Setup** window for the selected pins.

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



Fields in the main grid-display window are not editable, but instead show the status of what you configure using the **PowerSetup** fields at the bottom of the window. You can select multiple rows and click the **Power Setup** fields to make multiple changes to many pins simultaneously.

- **PowerType** can either be **Buffer Connection**, **Reference Pin**, or **Power Model**.

  — **Buffer Connection** means the pin are directly connected to the Pull Up or Pull Down, according to what was specified in the Signal Setup window.

  — **Reference Pin** does not specifically assign a Power source or Ground, but this setting can be used for pin mapping.

  — **Power Model** allows a one or two port component to be connected directly to the pin. Click the **[...]** button to load a model. If the component selected for the Power Model has two pins, the **Pin Mapping** area is displayed. The top **Pin Mapping** area selects which of the Power Model's terminals is directly connected to the Power Pin selected in the pin grid. The bottom **Pin Mapping** area allows you to assign the remaining Power Model terminal to a net.

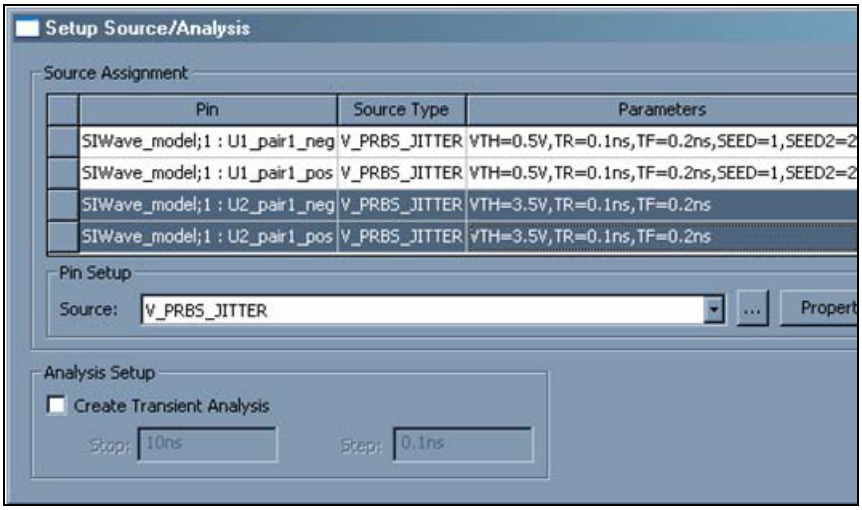
- Clicking **Properties** allows you to customize the parameters for the selected Power Model component.
- In order to connect a ground terminal, select Power Type: **Power Model**, then click the **[...]** button to load a **RES_ component** as the Power Model. This two port component, when not configured with a resistance parameter, acts as a short. The pin mapping for the **RES_ Power Component** allows you to then short the Power pin to Ground.

Click **OK** to close the window and implement any changes that have been configured.

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

IBIS I/O Wizard Source/Analysis Setup

To configure source assignments, click **Setup Sources/Analysis** in the I/O Wizard window; this opens the **Setup Source/Analysis** window for the pins designated as “Signal”.



This window allows users to add sources to the project. Sources are connected to the selected drivers that have been configured for each of the signal pins. A transient analysis can also be set up, which streamlines the process. The fields in the window reflect customizations made to the Sources/Probe pin which allows for toggling the existence of a probe for the IBIS buffer.

- By default, a PRBS source with jitter is assigned to each signal pin. Sources are attached to the logic pin on the IBIS buffer (if applicable), not the actual signal pin. The **Source** selection box lists only the independent sources in the project. Grounds are automatically added to the other side of the source.

Click the **[...]** button to load another source , then click Properties to set the per-instance properties for the source. Checking the Probe box attaches a probe directly to the selected pin.

From the **Analysis Setup** area, set up a quick transient analysis by checking **Create Transient Analysis**, which enables the **Stop** and **Step** group boxes for controlling the analysis. After the window is closed, the setup can be further refined on the **Project Tree**.

Click **OK** to close the window and implement any changes that have been configured.

AMI Analysis

The IBIS Algorithmic Modeling Interface (AMI) allows time-domain simulation of a linear-channel Circuit design using customer-supplied models for the transmitter and receiver. Reports include plots of step responses from VerifEye, eye diagrams like those available for QuickEye, and bit-error-rate (BER) contours.

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

Set Up and Run AMI Analysis

AMI analysis is equivalent to the Time Domain simulation described in the IBIS AMI specification. To run AMI analysis, the design must include one or more IBIS AMI transmitters (or AMI Sources) and corresponding IBIS AMI receivers (or AMI Probes). Transmitter and receiver parameters are set up in the sources and probes (See Importing the AMI Transmitter and Receiver).

Other parameters and options can be specified in the AMI Analysis setup window, as described in the following topics.

AMI Analysis Outputs

To open the results of an AMI analysis, click the Results icon in the **Project Manager** window and select one of the following **Create Report** options.

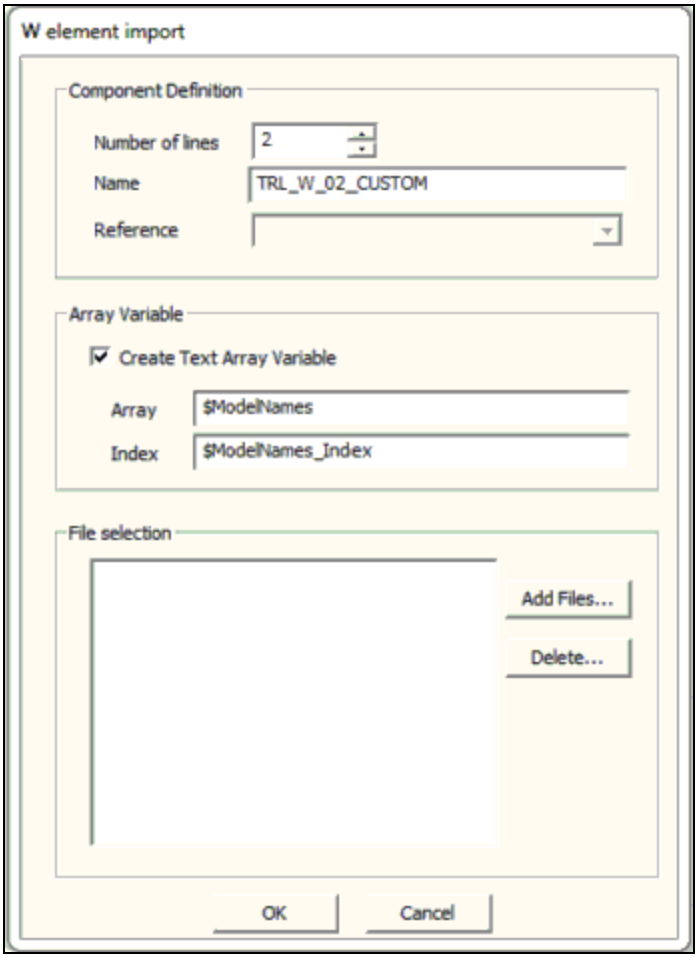
W-Element Model Support

Nexxim supports the import of W-element models via an import wizard.

To open the wizard:

1. View the **Component Libraries**. Open the **Symbols** area. Click **Import Models**.
2. From the area, select the **W-ELT** icon to open the **W element import** 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. Enter the **Number of lines** for the W-element.

4. Click **Add Files** to browse to a directory containing the data files for the W-element.

You can select multiple files. If **Create Text Array Variable** is checked, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform creates a text array index to store the multiple files. The model is then created as a parametric model.

5. Click **OK**. An instance of the W-element is attached to the cursor, so drag it into the schematic to place the W-element. When instances of the component are placed, right-click and select **Finish** or press **Esc** on the keyboard to stop placing instances.

If the model was created as a parametric model, simulation setups can sweep the entries in the text array index (**$ModelNames_Index** in the example).

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

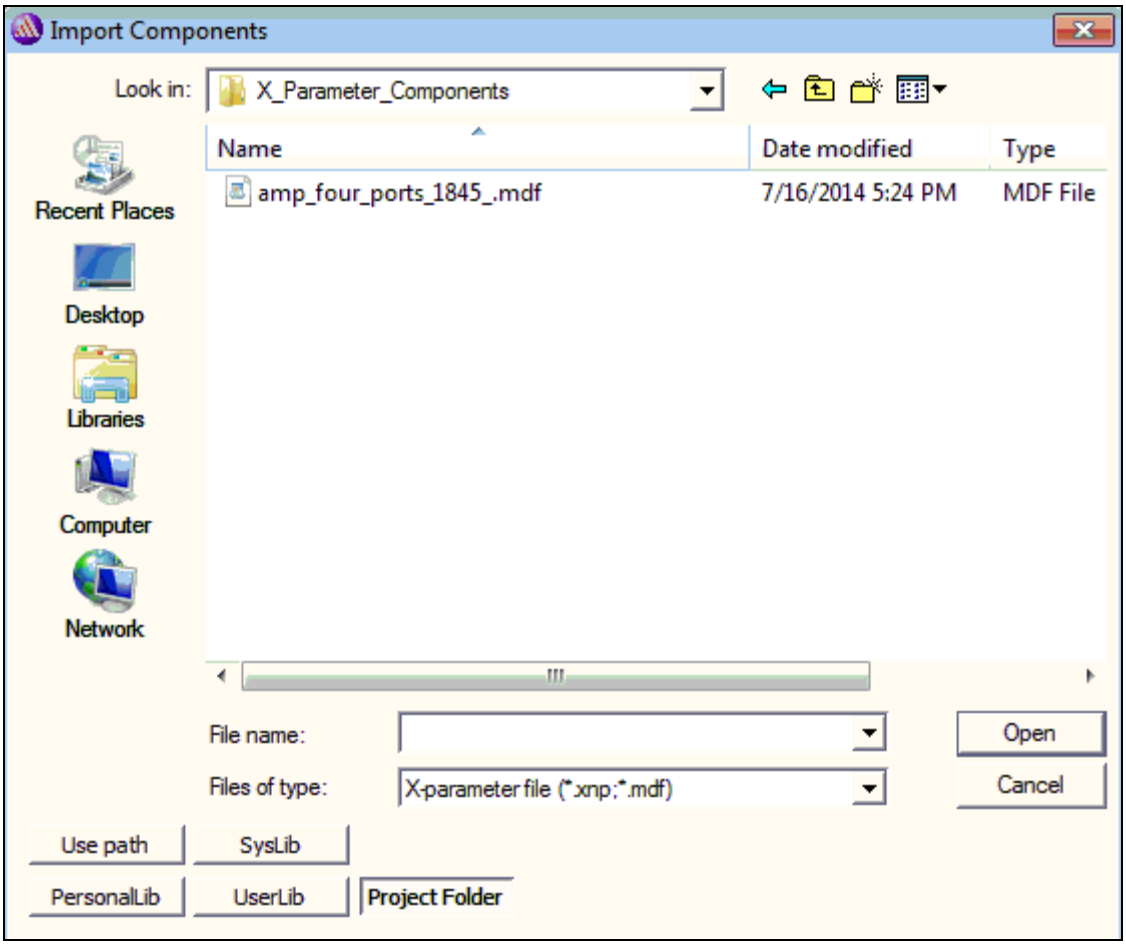
X-Parameter Model Support

X-parameters are the nonlinear counterparts of S-parameters. They can be used to model the frequency-domain behavior of driven nonlinear systems such as amplifiers and mixers.

Importing an X-Parameter Component

Nexxim supports the import of X-parameter models via an import wizard. To open the wizard,

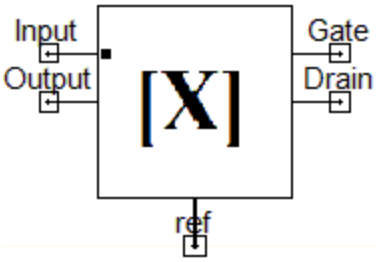
1. View the **Component Libraries**. Open the **Symbols** area. Click **Import Models**.
2. From the area of icons, select the **XParam** icon to open the **Import Components** window.
3. Select the file containing the X-parameter component information.



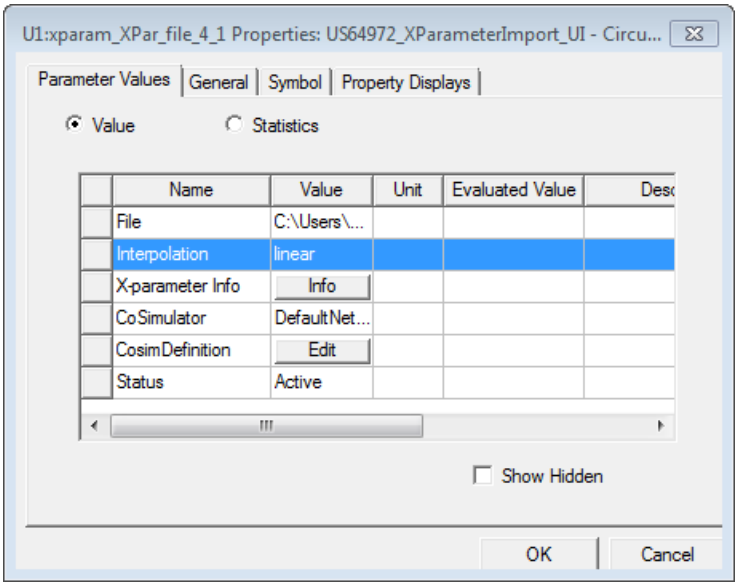
4. Select the file containing the X-parameter component information.
5. Click **Open** and an instance of the X-parameter element attaches to the cursor. Drag and

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

place it in the schematic. The port configuration is read from the file.



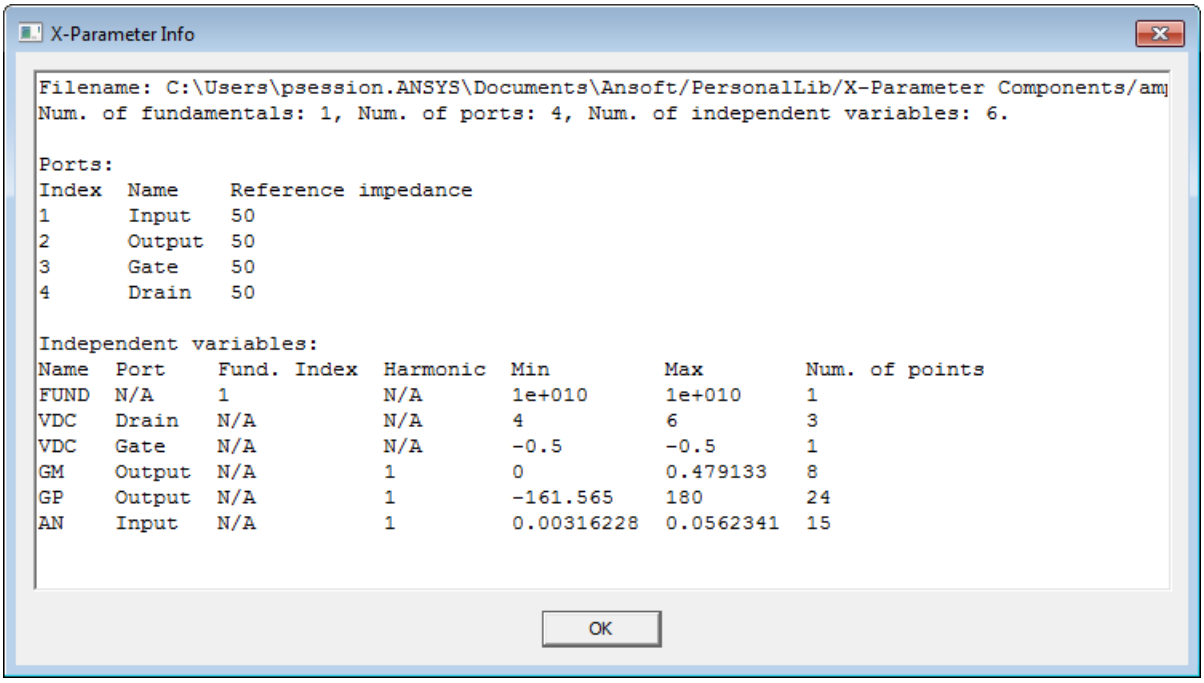
6. When you are finished placing instances of the component, right-click and select **Finish** or press **Esc** to stop placing instances.
7. Select the component to display its **Properties**.



The **Interpolation** property: Linear interpolation is the only choice currently available for calculating X-parameter values for solution points that are not in the file data.

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 **X-parameter Info** to display a (read-only) summary of the contents of the file.



The **Info** window displays:

From the top lines:

- The name of the file.
- The number of fundamental large-signal inputs.
- The number of ports.
- The number of independent variables in the file.

For each of the Ports:

- The port number.
- The port name.
- The reference impedance.

For each of the Independent Variables:

- The name of the variable.
- For the FUND variable, the index of that fundamental. A FUND with index 1 is required.
- For the VDC and IDC variables, the name of the port corresponding to the port number given in the file.
- For the variables after FUND, VDC and IDC, the name of the port and the harmonic number.

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

- For all variables, the minimum and maximum values used in generating the X-parameters.
- For all variables, the number of sweep points employed in generating the data file. The number of sweep points affects the accuracy of the calculation when interpolation is required.

See Independent Variables for X-Parameters for details.

9. Click **OK** to close the X-Parameter Info window.
10. Click **OK** to close the Properties window.
11. Connect the X-parameter component to the circuit. The X-parameter model can be simulated only with Nexxim Harmonic Balance analysis. See the X-Parameter Technical Notes for information on the implementation.

X-Parameter Technical Notes

X-parameters are a superset of the frequency-dependent S-parameters. X-parameters are intended to model a device that generates a nonlinear response to an input whose energy is concentrated at one or a few fundamental frequencies. The response at each port consists of signals at the harmonics or the fundamental, or at the sums and differences of the harmonic frequencies of the fundamentals when more than one is present. The treatment of X-parameter models depends on the principle of *harmonic superposition*—the idea the harmonics are small in magnitude relative to the fundamental signal, and thus the harmonics can be superimposed linearly.

The X-parameter solver combines large-signal and small-signal models. In systems with nonlinear components, large-signal AC magnitudes can affect the operating point; small-signal contributors can be modeled accurately by linearizing the circuit around an operating point. The X-parameter solver calculates the Large-Scale Operating Point (LSOP) based on the large-signal contributors. The small-signal harmonic contributions are obtained by linearizing the system around the LSOP.

Let a be the vector of complex incident wave amplitudes to the device. Each element of the vector is indexed by port (1 to P) and harmonic (1 to N).

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

$$a = \begin{bmatrix} a_{11} \\ \cdots \\ a_{1N} \\ a_{21} \\ \cdots \\ a_{2N} \\ \cdots \\ a_{P1} \\ \cdots \\ a_{PN} \end{bmatrix}$$

(1)

Vector *b* is a similarly indexed vector of scattered wave amplitudes. The vector *b* is a nonlinear complex function of *a*:

$$b = f(a)$$

(2)

From the situation represented by an X-parameter element, vector *a* is composed of one or a few large signals and several small-signal harmonics:

$$a = a_L + a_S$$

(3)

where the vector a_L contains the complex amplitudes of the large signals and zeros instead of the small-signal amplitudes. Similarly, the vector a_S contains the complex amplitudes of the small signals and zeros instead of the large-signal amplitudes.

If the *a* inputs are real, you could approximate *b* as:

$$b = f(a_L) + J(a_L)a_S$$

(4)

where $J(a_L)$ is a Jacobian matrix of the derivatives of *f* at $a = a_L$ with respect to the small-signals a_S.

But *a* and *b* are complex, so a different formulation is appropriate.

Start by splitting *a* into its real and imaginary parts:

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

$$a = a^r + ja^i$$

(5)

$j = \sqrt{-1}$ where , and the superscripts *r* and *i* denote the real and imaginary parts of *a*, not exponents. Consider *f* as a function of two real vectors:

$$f(a) = f(a^r, a^i)$$

(6)

Then, linearizing *f* around the LSOP, you have:

$$f(a^r, a^i) \cong f(a_L{}^r, a_L{}^i) + \frac{\partial f}{\partial a_S{}^r}(a_S{}^r) + \frac{\partial f}{\partial a_S{}^i}(a_S{}^i)$$

(7)

Use the following identities to rewrite (7) in terms of a_S and its complex conjugate a_S^*:

$$a_S{}^r = \frac{a_S + a_S{}^*}{2}$$

(8)

$$a_S{}^i = \frac{a_S - a_S{}^*}{2j}$$

(9)

Substituting (8) and (9) into (7), obtaining:

$$f(a) = f(a_L{}^r, a_L{}^i) + \frac{\partial f}{\partial a^r}\left(\frac{a_S + a_S{}^*}{2}\right) + \frac{\partial f}{\partial a^i}\left(\frac{a_S - a_S{}^*}{2j}\right)$$

(10)

Rearranging the terms involvinga_S and its complex conjugate a_S^*:

$$f(a) \cong f(a_L{}^r, a_L{}^i) + \frac{1}{2}\left(\frac{\partial f}{\partial a^r} + \frac{1}{j}\frac{\partial f}{\partial a^i}\right)a_S + \frac{1}{2}\left(\frac{\partial f}{\partial a^r} - \frac{1}{j}\frac{\partial f}{\partial a^i}\right)a_S{}^*$$

(11)

Further simplify equation (11) to show the parallels with equation (4). Let:

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

$$b_0 = f(a_L^r, a_L^i)$$

(12)

$$S = \frac{1}{2}\left(\frac{\partial f}{\partial a^r} + \frac{1}{j}\frac{\partial f}{\partial a^i}\right)$$

(13)

$$T = \frac{1}{2}\left(\frac{\partial f}{\partial a^r} - \frac{1}{j}\frac{\partial f}{\partial a^i}\right)$$

(14)

The complex equivalent of equation (4) is the formula for calculating the response from a set of X-parameter data and incident waves:

$$b = b_0(a_L) + S(a_L)a_S + T(a_L)a_S^*$$

(15)

In words, b_0 is a function of the large-signal inputs, or more generally, of the LSOP, while $S(a_L)$ and $T(a_L)$ are matrices that represent the linearization of the small signal harmonics around the LSOP. The complex conjugate a_S^* is required to account for the phase differences between the harmonic components and the large signals.

When $||b_0|| = 0$, T=0and $b = S(a)$. Thus, X-parameters are a superset of S-parameters.

For each fundamental there is at least one large signal, and the LSOP includes just the absolute value of this signal. The dependence on phase of the large signal is determined on the time-invariance of the system. This dependence is captured by the use of three matrices, P_{b0}, P_S, and P_T, the entries of which are all the form $e^{jq\phi}$, with q an integer and ϕ the phase of a_L:

$$a_L = |a_L|e^{j\phi}$$

(16)

Using these matrices, the dependence of b_0, S, and T on a_L is written as:

$$b_0(a_L) = P_{b_m} b_0(|a_L|)$$

(17)

$$S(a_L) = S(|a_L|)P_S$$

(18)

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

$$T(a_L) = T(|a_L|)P_T$$

(19)

The LSOP can also depend on *independent variables* supplied in the X-parameter file. The independent variables include biasing and load conditions (See Independent Variables for X-Parameters). Then, b_0, S, and T all depend on the independent variables that are used when the X-parameter data file was generated:

$$b_0 = b_0(x)$$

(20)

$$S = S(x)$$

(21)

$$T = T(x)$$

(22)

where $x$is a vector of the M independent variables

$$x = \begin{bmatrix} x_1 \\ x_2 \\ \cdots \\ x_M \end{bmatrix}$$

(23)

DC Equations

Equation (15) relates the scattered wave amplitudes to the incident wave amplitudes, at the fundamental frequency and at higher harmonics. However, nonlinear devices typically exhibit some degree of rectification, which means that even when the input is a single tone, the output includes a DC component.

There are two options for modeling the DC response. The X-parameter model approximates either the DC voltage at a port given the current going into it, or the DC current at a port given the voltage across it. From the first case, the port is designated as an IDC port, and selected to bias the port by an external current source. From the second case, the port is designated as a VDC port, and selected to bias the port by an external voltage source.

For both cases, follow a similar procedure leading to equation (15), except alter the approximated functions (i.e., the DC currents, i_{DC}, or the DC voltages, v_{DC}).

For the IDC ports,

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

$$v_{DC} = v_0(x) + 0.5Z(x)a_S + 0.5Z^*(x)a_S^*$$
(24)

where v_{DC} is a vector of the approximate DC voltages at all IDC ports. The vector $v_0(x)$ denotes the DC voltages at the LSOP, and the harmonic impedance matrix $Z(x)$ relates the small-signal amplitudes at all harmonics to the DC voltages. The LSOP must include the DC currents at all the IDC ports. For the VDC ports,

$$i_{DC} = i_0(x) + 0.5Y(x)a_S + 0.5Y^*(x)a_S^*$$
(25)

where i_{DC} is a vector of the approximate DC currents at all VDC ports. The vector $i_0(x)$ denotes the DC currents at the LSOP, and the harmonic admittance matrix $Y(x)$ relates the small-signal amplitudes at all harmonics to the DC currents. The LSOP must include the DC voltages at all the VDC ports.

What should be written is (24) and (25) in the same form as (15), but they can be written more succinctly as:

$$v_{DC} = v_0(x) + \mathrm{Re}(Z(x)a_S)$$
(26)

and

$$i_{DC} = i_0(x) + \mathrm{Re}(Y(x)a_S)$$
(27)

which also make it clear that v_{DC} and i_{DC} are real.

Independent Variables for X-Parameters

The independent variables in the X-parameter file are real values, indexed by fundamental, port, and harmonic numbers.

X-Parameter Independent Variables

Independent Variable	Description
fund_k	k^{th} fundamental frequency.
VDC_p	DC voltage applied to port *p*. Mutually exclusive with IDC_p at the same port.
IDC_p	DC current applied to port *p*. Mutually exclusive with VDC_p at the same port.
AN_p_n	Magnitude of Reference large-signal input to port p at harmonic n. One AN variable must be present for each fundamental.

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

Independent Variable	Description
AM_p_n	Magnitude of any other large-signal input to port p at harmonic n other than the Reference Signal AN_p_n. Required when AP_p_n is used at same port and harmonic.
AP_p_n	Phase in degrees of any other large-signal input to port p at harmonic n other than the Reference Signal AN_p_n. Required when AM_p_n is used at same port and harmonic.
GM_p_n	Magnitude of the reflection coefficient of the load at port p, harmonic n. Required when GP_p_n is used at the same port and harmonic. Mutually exclusive with the other formats for specifying load at the same port p and harmonic n.
GP_p_n	Phase in degrees of the reflection coefficient of the load at port p, harmonic n. Required when GM_p_n is used at the same port and harmonic. Mutually exclusive with the other formats for specifying load at the same port p and harmonic n.
GX_p_n **GY_p_n**	Real and imaginary parts of the reflection coefficient. Mutually exclusive with the other formats for specifying load at the same port p and harmonic n.
ZM_p_n **ZP_p_n**	Magnitude and phase of the load impedance. Mutually exclusive with the other formats for specifying load at the same port p and harmonic n.
ZX_p_n **ZY_p_n**	Real and imaginary parts of the load impedance. Mutually exclusive with the other formats for specifying load at the same port p and harmonic n.

X-Parameter References

[1] David E. Root, Jan Verspecht, David Sharrit, John Wood, and Alex Cognata, “Broad-Band Poly-Harmonic Distortion (PHD) Behavioral Models From Fast Automated Simulations and Large-Signal Vectorial Network Measurements”, *IEEE Transactions on Microwave Theory and Techniques*, Vol 53, No. 11, November 2005, pp. 3656-3664.

[2] Jan Verspecht and David E. Root, “Polyharmonic Distortion Modeling”, *IEEE Microwave Magazine*, Vol. 7, Issue 3, June 2006, pp. 44-57.

State Space Model Support

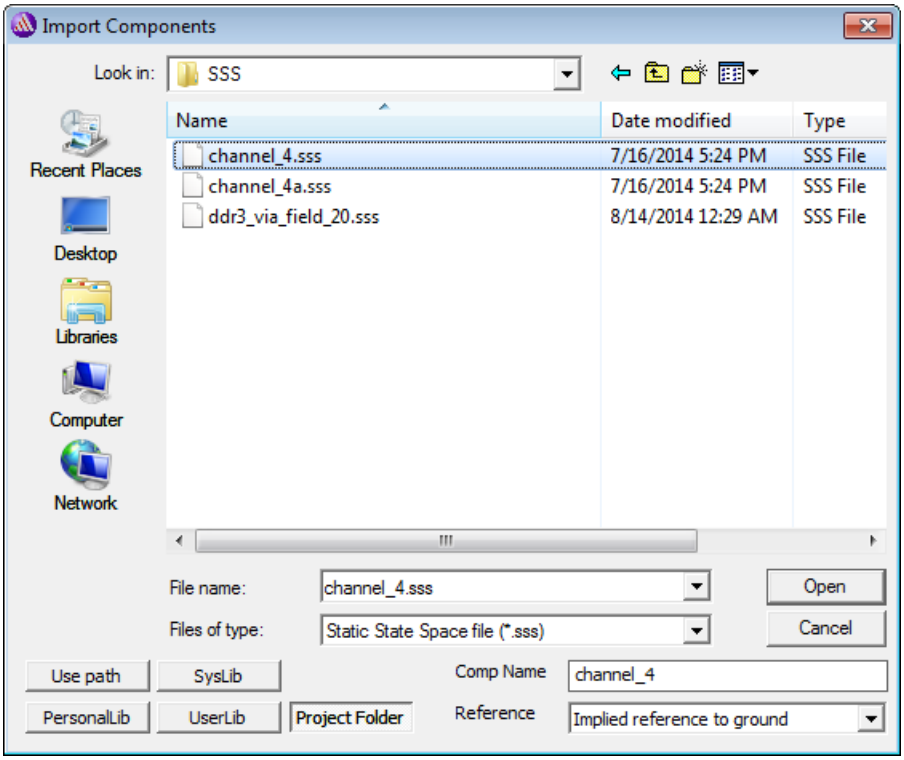
A state-space model is a representation of frequency-dependent transfer function data. Passivity and causality enforcement work best with state-space models. You can create a state-space model from a set of SYZ-parameters via the Network Data Explorer. See State Space Method in the Circuit Design Technical Notes for details.

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

Importing State-Space Models

Once the state-space data have been generated and saved, you can import it just like any other static model.

1. View the **Component Libraries**. Open the **Symbols** area. Click **Import Models**. From the area, select the **SSS** icon to open the **Import Components** window.



2. Select the file containing the state-space data.
3. In **Comp Name** in the lower-right, change the component name if selected.
4. Select a type for the **Reference** node:
   - **Implied reference to ground**: the global ground is the reference port for each N port connection. Its voltage is always 0V.
   - **Common reference port**: an additional reference pin is added that is a common reference for the definition of port voltages. This reference can have any voltage and corresponds to the net for that extra pin.
   - **Add individual hidden reference pin per port**: an additional reference pin is added for each port. These reference pins are hidden, set to 0V by default, and can each have their own reference voltage. You can connect each reference pin to its own net. Select this option if you do not want to see the reference pins.

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

- **Add individual reference pin per port**: an additional reference pin is added for each port. Every reference pin can have its own reference voltage and be connected to its own net.

> **Note:**
>
> See Reference Nodes on S-Parameter Elements in the S-parameter Technical Notes for how the Circuit solver handles reference node configurations.

5. Click **Open**. An instance of the state-space N-port component is attached to the cursor, so drag and place it in the schematic. When you are finished placing instances of the component, right-click and select **Finish** or press **Esc** on the keyboard to stop placing instances.

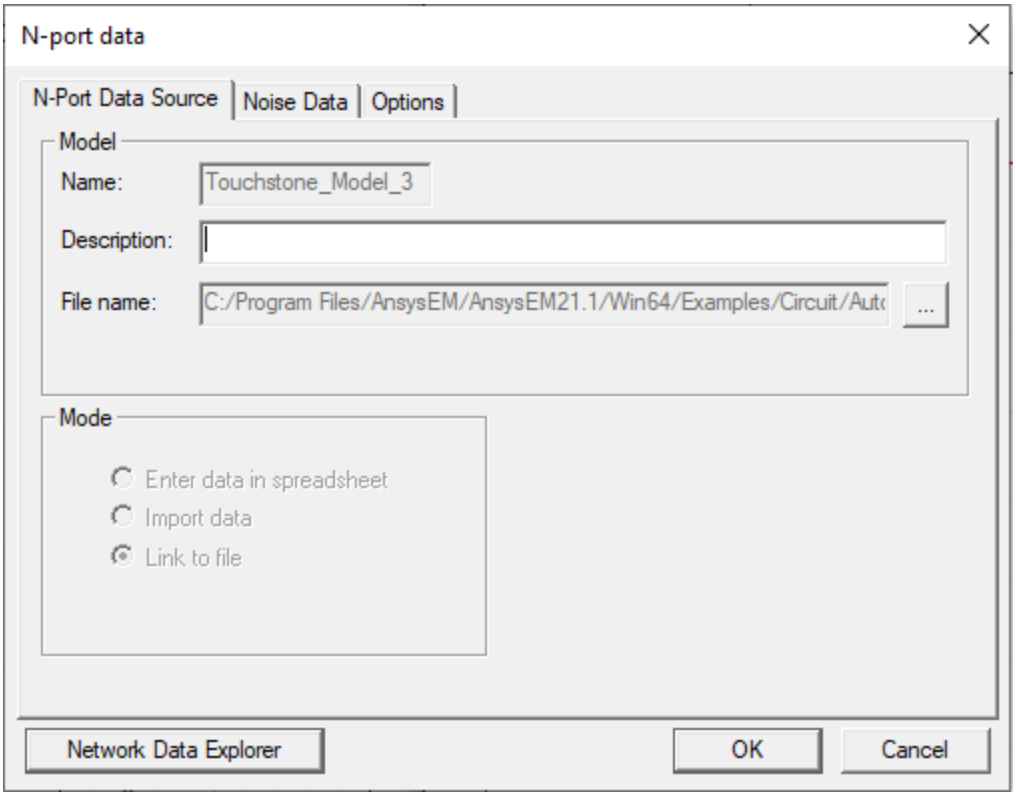
Editing a State Space Model

To view the details of the model, right-click the state space component in the schematic and select **Edit Model** to open the **N-port data** window. The **N-port data** window allows you to view or in some cases to change the name, number of ports, characterizing data, and other properties for the N-port.

State Space N-Port Data Source Tab

The **N-Port Data Source** tab is used to name the N-port device and specify the source for the data, typically an external file.

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



- **Name** — Displays the name of the imported solution as it appears in the **Project Tree**.
- **Description** – Used to set or change the description of the imported solution. To change the description, click in the box , then type the description you want.
- **File Name** – Used to display or change the file name of the imported solution. To change the file name, browse (...) for a file.
- **Mode** – The network solution data is read from an external **Link to file** at analysis time.
- **Network Data Explorer** allows you to view solution data by opening the Network Data Explorer.

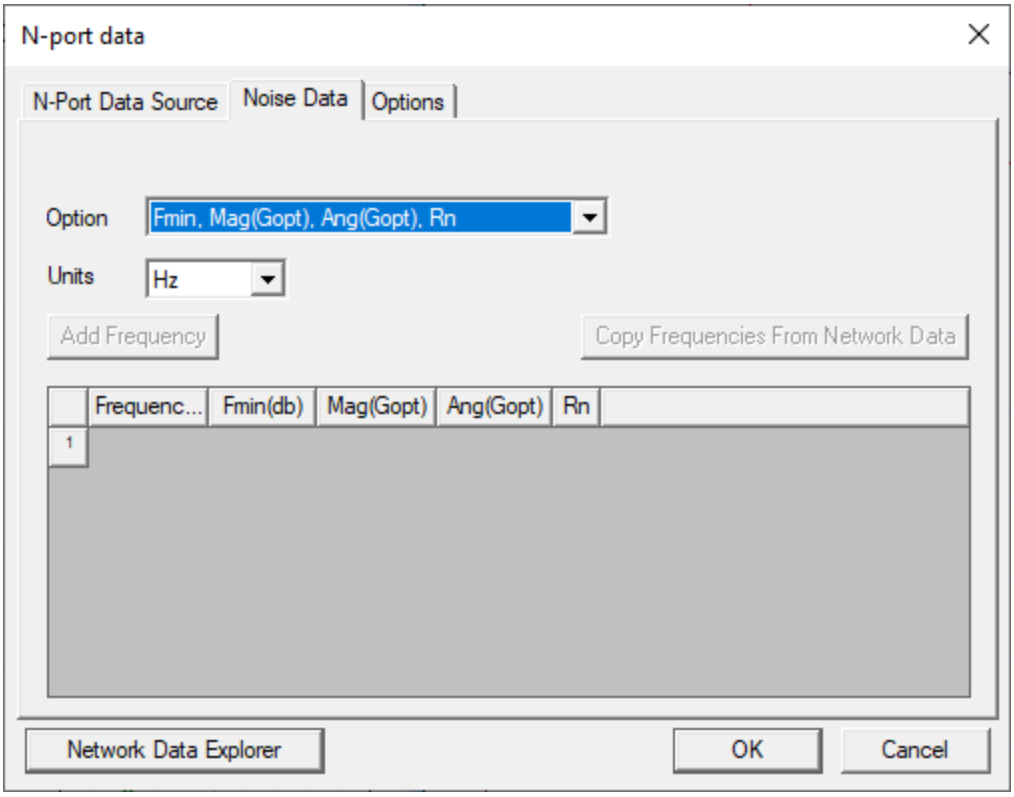
> **Note:**
>
> **Network Data Explorer** is not activated for State-Space models.

When the **N-Port Data Source** options have been entered, click **OK** or select another tab.

State Space N-Port Data Noise Data Tab

Click the **Noise Data** tab to enter frequency-dependent noise parameters.

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



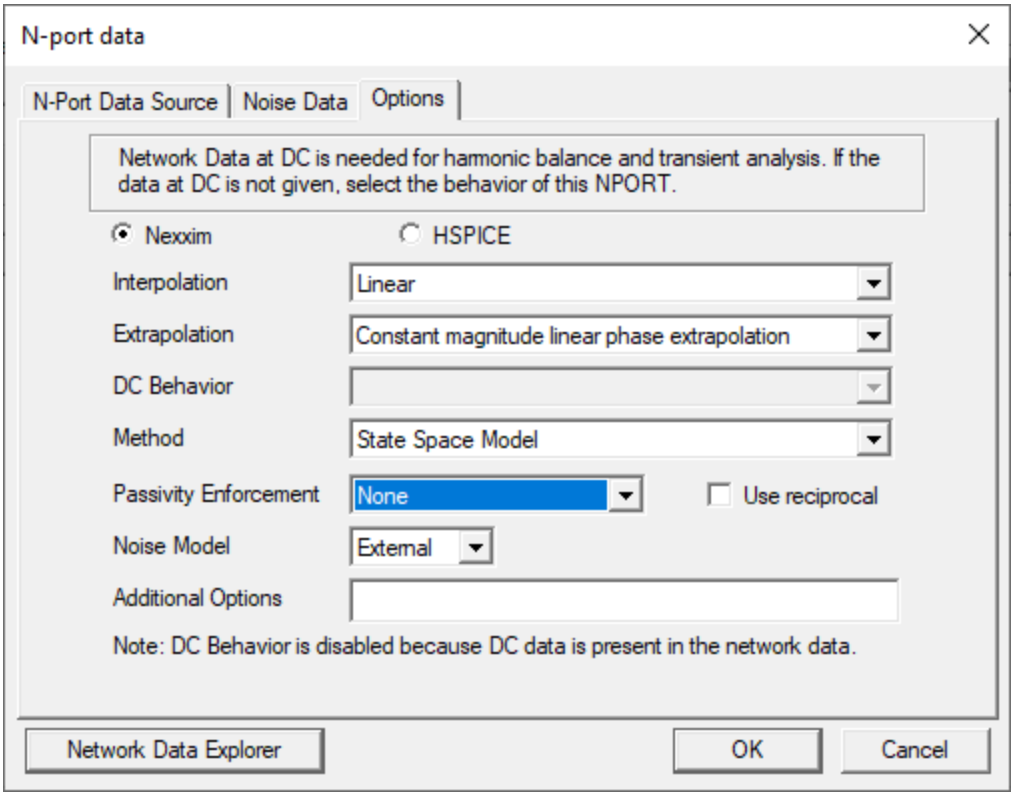
- The **Option** field selects the units for the noise parameters:
- The **Units** field selects the units of frequency: **Hz**, **KHz**, **MHz**, **GHz**, **THz**, or **rps**.
- click **Copy Frequencies from Network Data** to create a list of frequencies that correspond to the frequencies in the N-port data file.
- click **Add Frequency** to add a new frequency to the end of the list.

Enter the noise data for each frequency in the list, then click **OK** or select another tab.

State Space N-Port Data Options Tab

The **Options** tab is used to select the behavior of the selected NPort.

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



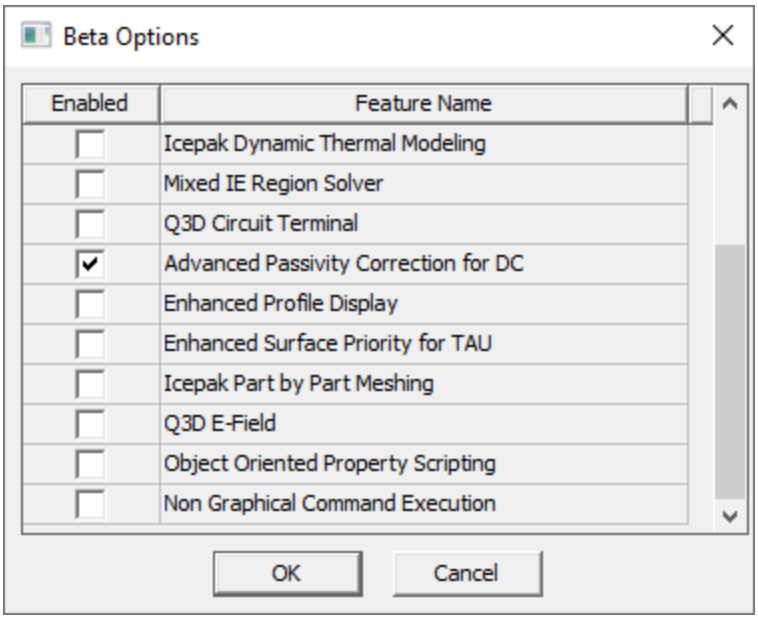
The following controls are available:

- **Nexxim/HSPICE** — Sets the NPort type. The selections for Interpolation, Extrapolation, DC Behavior, Method, and Passivity Enforcement depend on the NPort type selected.
- **Interpolation** — Specifies the interpolation method for data points in the frequency range of the Touchstone data.
- **Extrapolation** – Specifies the extrapolation method for data points outside the frequency range of the Touchstone data.
- **DC Behavior** — Specifies behavior of the N-Port at DC (zero frequency).
- **Method** – Specifies the method used to convert frequency domain data to the time domain. Choice of methods is available only for Nexxim Nports.
- **Passivity Enforcement** — Performs a passivity check on the S-parameter data and attempts to correct non-passive data. The options are None, IFPV, IFPVLF, Convex Optimization, and Passivity by Perturbation.

  Passive devices may dissipate or temporarily store energy, but never generate energy. Use the drop-down menu to select the method for enforcing passivity. Passivity enforcement is available only for Nexxim NPorts. See Passivity Checking and Enforcement. For details on passivity checking and enforcement, see S-Parameter Technical Notes.

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

When applying IFPVLF, the passive DC fit to Scattering Parameter Data (S-data) can deteriorate to the point the inaccuracies of the (passive) fit to the corresponding Impedance Data (Z-data) at zero frequency (DC) and low-frequency may cause inaccurate time-domain steady state simulations. If preserving an accurate passive DC fit to S-data is important, like it is in power electronics applications, e.g., with DC-DC converters and power supply Voltage Regulator Module (VRM)), then click the **Advanced Passivity Correction for DC** beta option to enhance the passive DC fit to S-data using IFPVLF. The **Advanced Passivity Correction for DC** is located in the **Beta Options** in the Global Options.



- **Use reciprocal** – Computes the inverse, or reciprocal, at each frequency so the N-port can be used for deembedding. See Deembedding S-Matrices.
- **Noise Model** – Specifies noise model for DC and frequency-domain analyses:
  - **None** – No noise analysis
  - **Internal** – Use internal noise model
  - **External** – Use noise data from Touchstone file if present, else use internal noise model
- **Additional Options** — Allows you to add model-level options for Nexxim or HSPICE.

When the N-port Options have been entered, click **OK** or select another tab.

- **Network Data Explorer** – Allows you to view solution data by opening the Network Data Explorer.

When the N-port Options have been entered, click **OK** or select another tab.

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

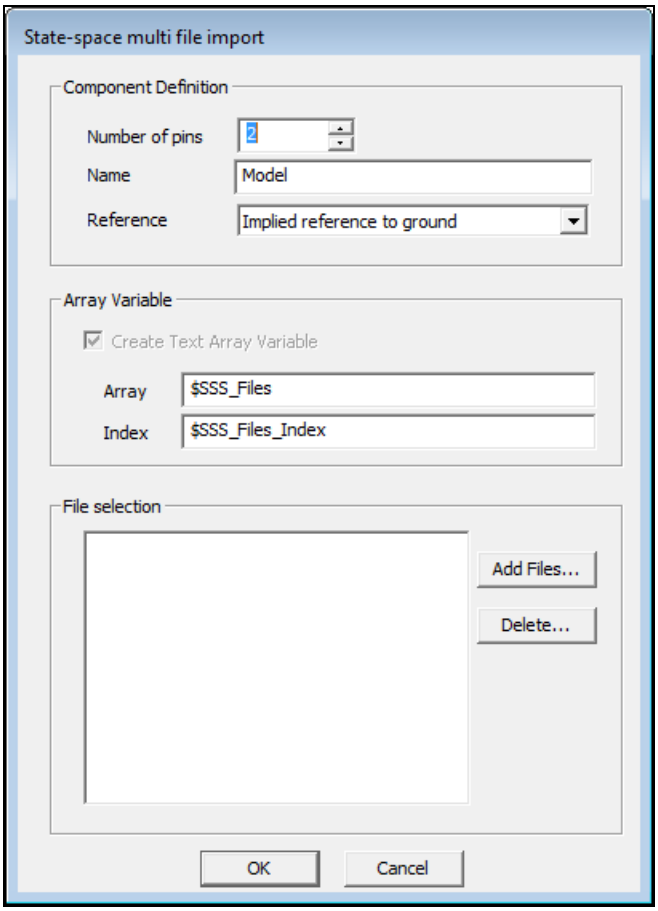
Editing State Space N-Port Symbol Pins

To edit the pins on the symbol for the N-Port component, right-click the component and select **Edit Symbol Bus Pins**. The procedure is identical for all flavors of N-Port. See Editing All N-Port Symbol Bus Pins for details.

Creating a Multi-File State Space Model

Instead of referencing a single file, an state-space model can reference an array of files, using a project array variable. For more information, see Adding a Project Variable Array. Follow these steps to create a multi-file model using array variables.

1. View the **Component Libraries** area, select the **Symbols** tab. Click **Import Models**.
2. Select the **SSS_Multi** icon. This opens the **State Space multi file import** window.



3. Enter a number of ports (from 1 to 1000). The number of ports determines what files select.
4. Enter a **Name** for the model if selected.

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 a type for the **Reference** node:
    - **Implied reference to ground**: the global ground is the reference port for each N port connection. Its voltage is always 0V.
    - **Common reference port**: an additional reference pin is added that is a common reference for the definition of port voltages. This reference can have any voltage and corresponds to the net for that extra pin.
    - **Add individual hidden reference pin per port**: an additional reference pin is added for each port. These reference pins are hidden, set to 0V by default, and can each have their own reference voltage. You can connect each reference pin to its own net. Select this option if you do not want to see the reference pins.
    - **Add individual reference pin per port**: an additional reference pin is added for each port. Every reference pin can have its own reference voltage and be connected to its own net.

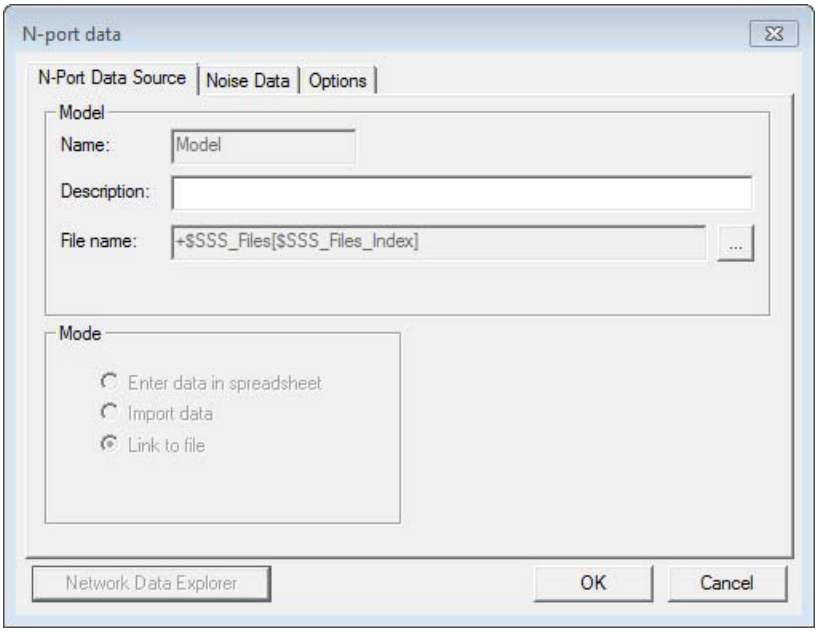
> **Note:**
>
> See Reference Nodes on S-Parameter Elements in the S-parameter Technical Notes for how the Circuit solver handles reference node configurations.

6. Click **Add Files** to browse and select multiple State Space (.sss) files to add to the list box.
7. Click **OK**. An instance of the state-space component is attached to the cursor, so drag and place it in the schematic. When instances of the component are placed, right-click and select **Finish** or press **Esc** on the keyboard to stop placing instances.
8. The simulation can sweep the entries in the created Text Array Variable. In this case, **$SSS_Files_Index** is the variable to sweep.

Editing a Multi-File State Space Model

To view or change the configuration of the N-Port model, right-click the component and select **Edit Model** on the menu. The **N-port data** window opens on its **N-Port Data Source** tab.

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 name** field are populated with the new text array variable expression.

See Editing a State Space Model for details on the **Noise Data** and **Options** tabs.

Importing Network Design Data (SYZ-Parameters)

N-ports are generic components for which characteristics may be defined in the form of user-supplied network-parameter data. They are called *black box* elements because no information regarding their function is provided except for their network parameters. The network parameters may be supplied as complex scattering parameters (*S* parameters), complex admittances (*Y* parameters), complex impedances (*Z* parameters), complex propagation constants (gamma parameters), or complex terminal characteristic impedances (Z_0 parameters). The data may be given in either polar form or in rectangular form.

Generally, the characterizing network parameters are frequency-dependent and is supplied for several frequency points. A common application of this capability is to introduce measured data into the simulation for comparison or modeling, or to represent a part of a circuit for which modeling is difficult or inappropriate. User-supplied network-parameter data may be entered in spreadsheet form or called from external files. This data then may be extracted on the data files and used in the simulation. Three data file formats are supported:

- **Touchstone/EESof Data** — This format uses the **.sNp, .yNp**, **.zNp**, **.tou**, or **.ts** file name extension. Only one dataset may be present in each Touchstone data file.
- **Circuit (Compact FLP)** – This format uses the **.flp** file name extension, and may contain several data groups, each of which is preceded by a header. The data group header identifies the group by means of a label, which is used as a reference name.

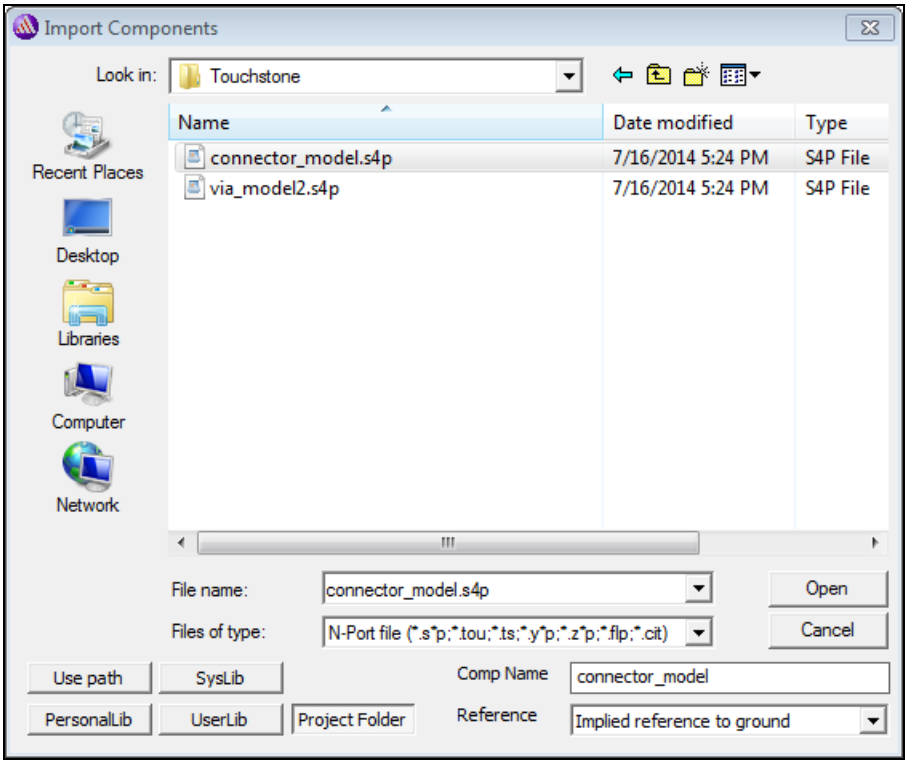
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **CITIfile** – This format uses the **.cit** file name extension, and contains S-parameter data in a proprietary format.

Creating an N-Port Model

An N-port is a component whose behavior is specified by a set of frequency-dependent data such as S-parameters. To create an N-port element to place, select the Circuit project that contains it.

1. View the **Component Libraries** area, select the **Symbols** tab. Click **Import Models**.
2. Select the **NPort** icon. The **Import Components** window opens.



3. Browse to the directory containing the N-port data file, and select the file.
4. In **Comp Name** in the lower-right, change the component name if selected.
5. Select a type for the **Reference** node:
   - **Implied reference to ground**: the global ground is the reference port for each N port connection. Its voltage is always 0V.
   - **Common reference port**: an additional reference pin is added that is a common reference for the definition of port voltages. This reference can have any voltage and corresponds to the net for that extra pin.

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

- **Add individual hidden reference pin per port**: an additional reference pin is added for each port. These reference pins are hidden, set to 0V by default, and can each have their own reference voltage. You can connect each reference pin to its own net. Select this option if you do not want to see the reference pins.
- **Add individual reference pin per port**: an additional reference pin is added for each port. Every reference pin can have its own reference voltage and be connected to its own net.

> **Note:**
>
> See Reference Nodes on S-Parameter Elements in the S-parameter Technical Notes for how the Circuit solver handles reference node configurations.

6. Click **Open** . An instance of the N-port component is attached to the cursor, so drag and place it in the schematic. When instances of the component are placed, right-click and select **Finish** or press **Esc** on the keyboard to stop placing instances.

Editing an N-Port Model

To view the details of the model, right-click the NPort component in the schematic and select **Edit Model** to open the **N-port data** window. The N-port data window allows you to view or in some cases to change the name, number of ports, characterizing data, and other properties for the N-port.

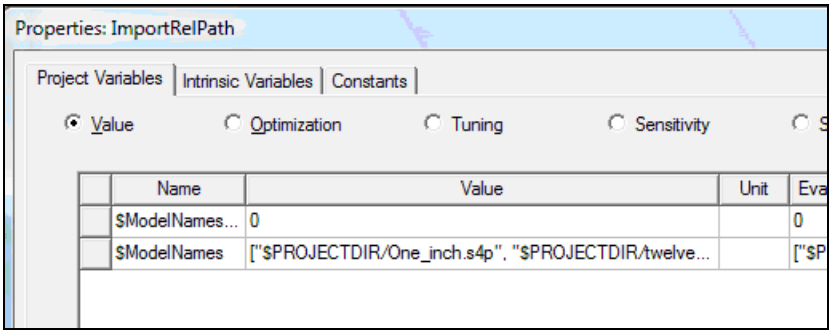
Creating a Multi-File N-Port Model

Instead of referencing a single file, an N-port model can reference an array of solution files, using a project array variable. For instance, the following can be used to specify the **File name**:

- +$ModelNames[1]
- +$ModelNames[$ModelNames_Index]

$ModelNames is a project level array variable and $ModelNames_Index is a project level array index variable.

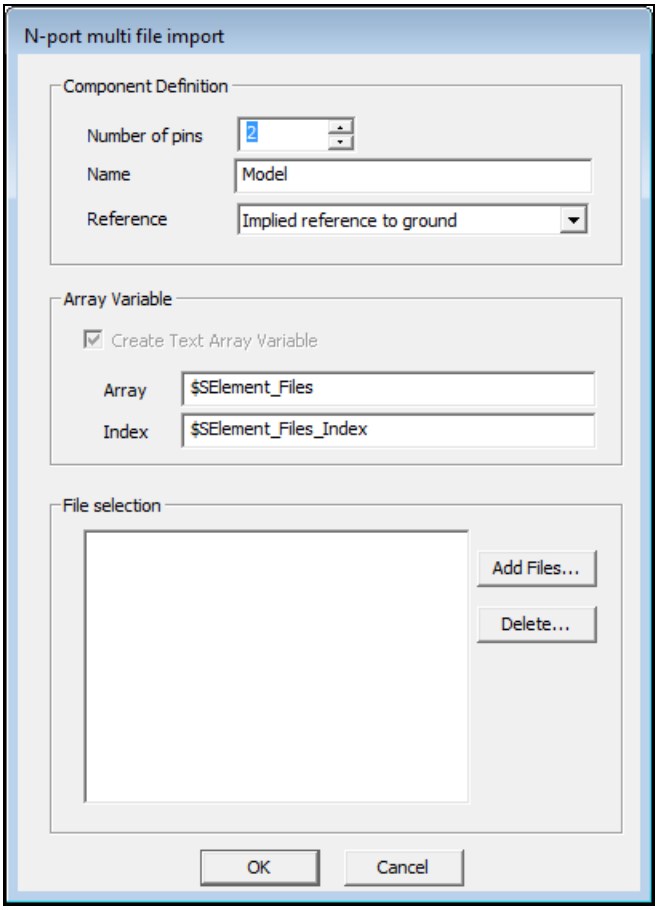
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For more information, see Adding a Project Variable Array.

To create a multi-file model using array variables:

1. View the **Component Libraries** area, select the **Symbols** tab. Click **Import Models**.
2. Select the **NPort_Multi** icon. This opens the **N-port multi file import** 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. Enter a number of pins or ports (from 1 to 1000). Files containing components with exactly the specified number of pins (and only those files) are displayed for selection in the **Add Files** window.
4. Enter a **Name** for the model if selected.
5. Select a type for the **Reference** node:
    - **Implied reference to ground**: the global ground is the reference port for each N port connection. Its voltage is always 0V.
    - **Common reference port**: an additional reference pin is added that is a common reference for the definition of port voltages. This reference can have any voltage and corresponds to the net for that extra pin.
    - **Add individual hidden reference pin per port**: an additional reference pin is added for each port. These reference pins are hidden, set to 0V by default, and can each have their own reference voltage. You can connect each reference pin to its own net. Select this option if you do not want to see the reference pins.
    - **Add individual reference pin per port**: an additional reference pin is added for each port. Every reference pin can have its own reference voltage and be connected to its own net.

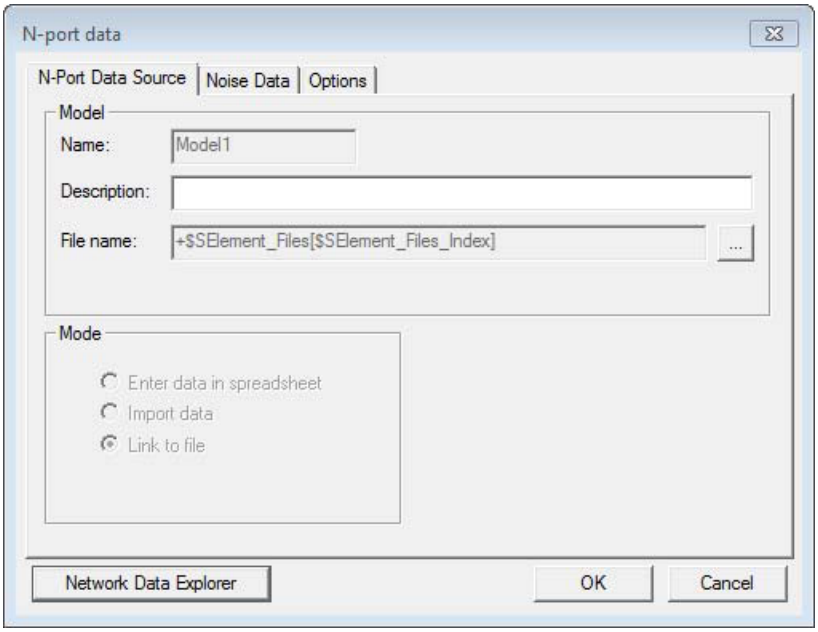
> **Note:**
>
> See Reference Nodes on S-Parameter Elements in the S-parameter Technical Notes for how the Circuit solver handles reference node configurations.

6. Clicking **Add Files** to browse and select multiple Touchstone (*.SnP) files to add to the list box.
7. Click **OK**. An instance of the N-port component is attached to the cursor, so drag and place it in the schematic. When instances of the component are placed, right-click and select **Finish** or press **Esc** on the keyboard to stop placing instances.
8. The simulation can sweep the entries in the created Text Array Variable. In this case, **$SElement_Files_Index** is the variable to sweep.

Editing a Multi-File N-Port Model

To view or change the configuration of the N-Port model, right-click the component and select **Edit Model** on the menu. The **N-port data** window opens on its **N-Port Data Source** tab.

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 name** field are populated with the new text array variable expression.

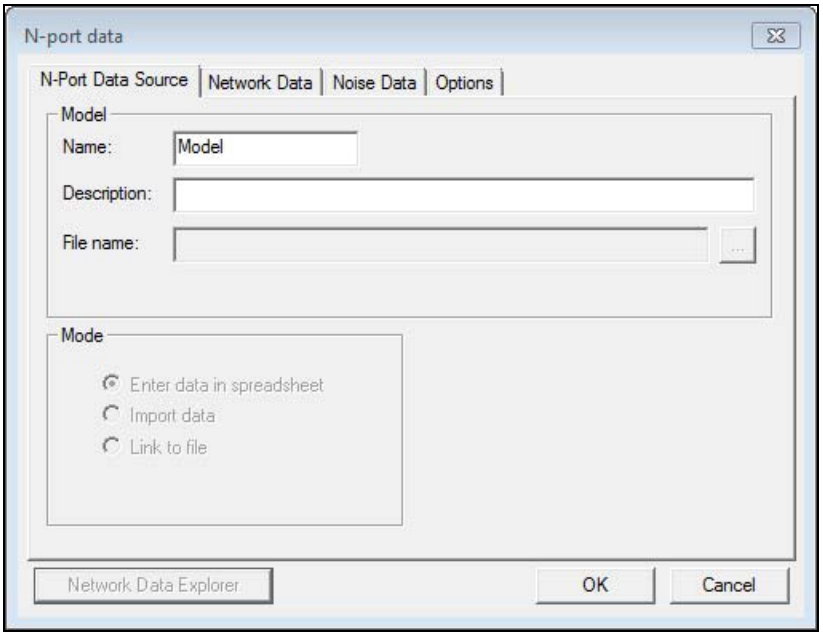
See Editing an N-Port Model for details on the **Noise Data** and **Options** tabs.

Creating an N-Port Model with No File

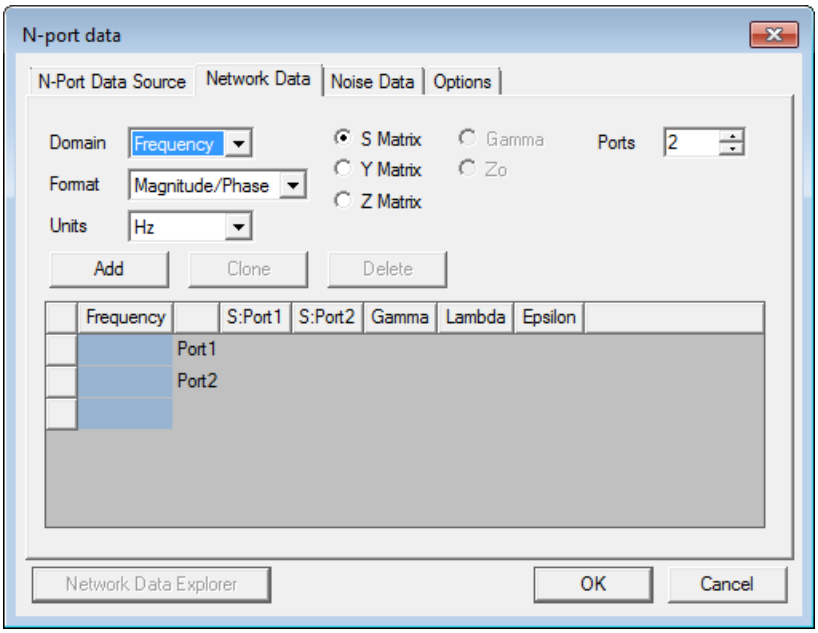
N-port frequency-dependent data such as S-parameters can be entered manually instead of reading it from a file. To create an N-port element to receive manually entered data, select the Circuit project that contains it.

1. View the **Component Libraries** area, select the **Symbols** tab. Click **Import Models**.
2. Select the **NPort_NoFile** icon to open the **N-port data** window. The **Mode** is set to **Enter data in spreadsheet**.

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 **Network Data** tab.



The **Network Data** tab is used to view the data from an imported file, or to enter the N-port data manually.

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

- **Domain** – Selects the **Frequency** or **Time** domain.
- **Format** – Selects the **Format** to one of the following: Magnitude/Phase, Real/Imaginary, or dB Phase.
- **Units** – From the Frequency domain, selects the units of frequency: **Hz**, **KHz**, **MHz**, **GHz**, **THz**, or **rps**. From the Time domain, selects the units of time: **fs**, **ps**, **ns**, **us**, **ms**, or **s**.
- **S Matrix** – Identifies the data as S-parameter values
- **Y Matrix** – Identifies the data as Y-parameter values
- **Z Matrix** – Identifies the data as Z-parameter values
- **Gamma** – Identifies the data as Gamma values
- **Z0** – Identifies the data as Z0 values
- **Ports** – Sets the number of signal ports
- **Phase Rotation** — Click to set the **Phase Rotation** of the **N-Port Network Data** source.The **Noise Data** and **Options** tabs are the same as for the file-based N-port. See Editing an N-Port Model.

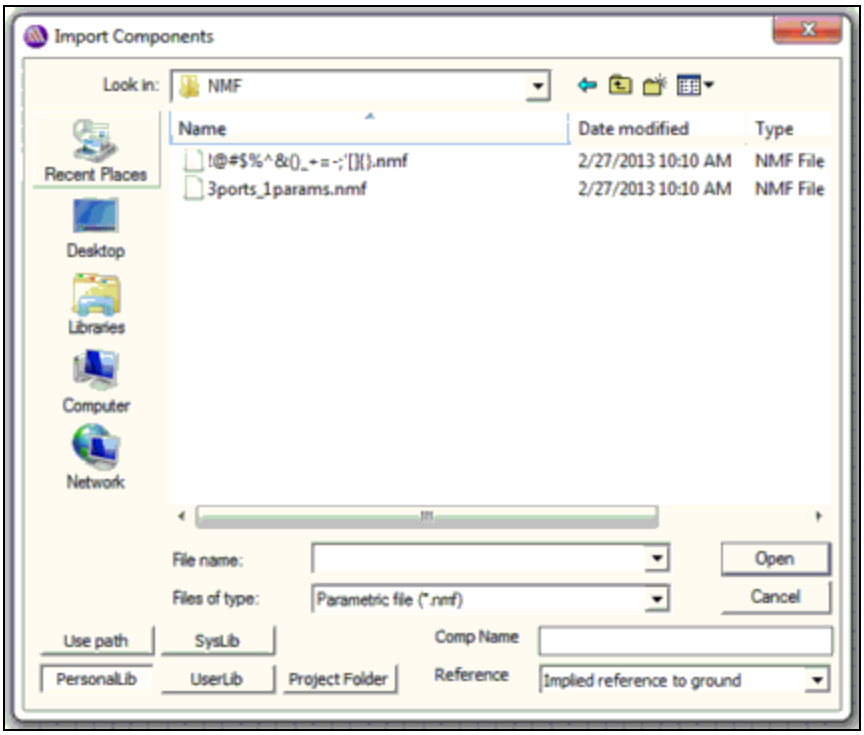
4. Click **OK**. An instance of the N-Port is attached to the cursor so drag and drop it in the schematic.

Creating a Parametric N-Port Model

A parametric N-port contains multiple values of key variables suitable for sweeping. To create a parametric N-port model:

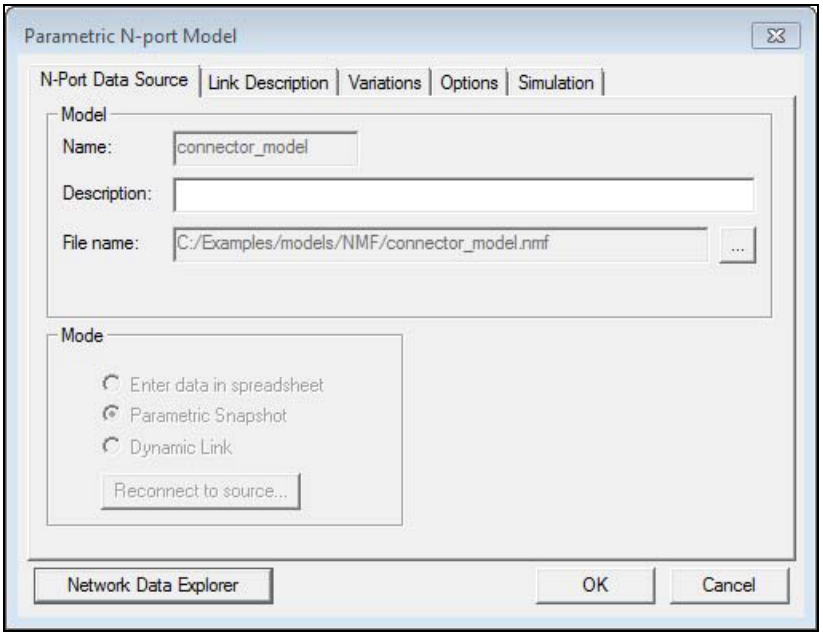
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. View the **Component Libraries** area, select the **Symbols** tab. Click **Import Models**.
2. Select the **NMF** icon. The **Import Components** window opens:



3. Browse to and select the file and click **Open**. An instance of the component is attached to the cursor so drag and drop it into the schematic. When you are finished placing instances of the component, right-click and select **Finish** or press **Esc** on the keyboard to stop placing instances.
4. Right-click the component and select **Edit Model** to open the **Parametric N-port Model** window.

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



The wizard has created the N-port in **Parametric snapshot** mode.

5. Click **OK**.

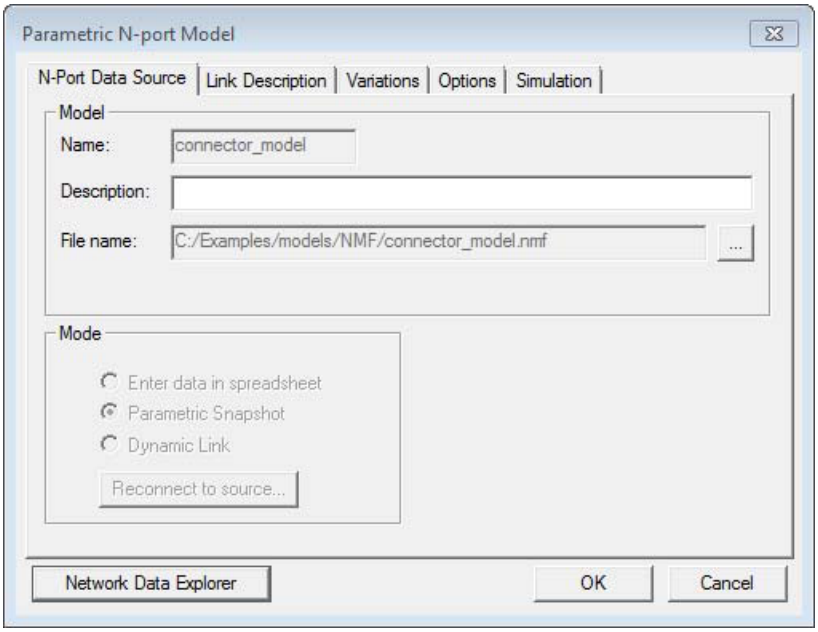
Editing a Parametric N-Port Model

From the **Parametric N-port Model** window, specify the basic characteristics of the Parametric *N*-port black boxes to place in a schematic or layout. To open the window, right-click the component and select **Edit Model** on the menu. The following tabs are available.

Parametric N-Port Data Source Tab

The **N-Port Data Source** tab is used to name the N-port device and specify the source for the data, typically an external file.

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



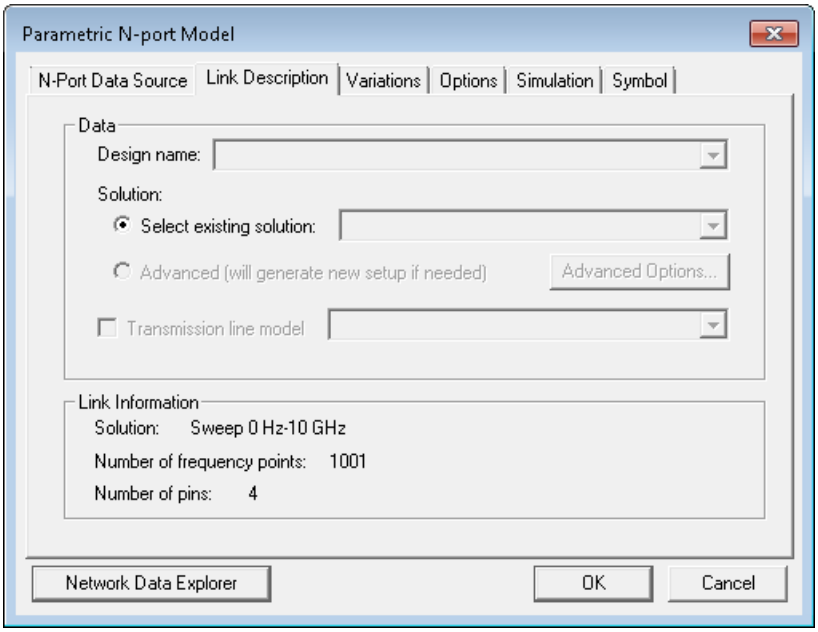
- **Name** — Displays the name of the imported solution as it appears in the **Project Tree**.
- **Description** – Used to set or change the description of the imported solution. To change the description, click in the box , then type the description you want.
- **File Name** – Displays the file name of the imported solution. To set the file name, click in the box , then type the file name you want, or browse (...) for a file.
- **Mode** – The network solution data has been copied into the project from an external file as a **Parametric Snapshot**.
- **Network Data Explorer** —Allows you to view and manipulate network data by opening theNetwork Data Explorer**.**

When the **N-Port Data Source** options have been entered, click **OK** or select another tab.

Parametric Link Description Tab

Click the **Link Description** tab to enter data and link information.

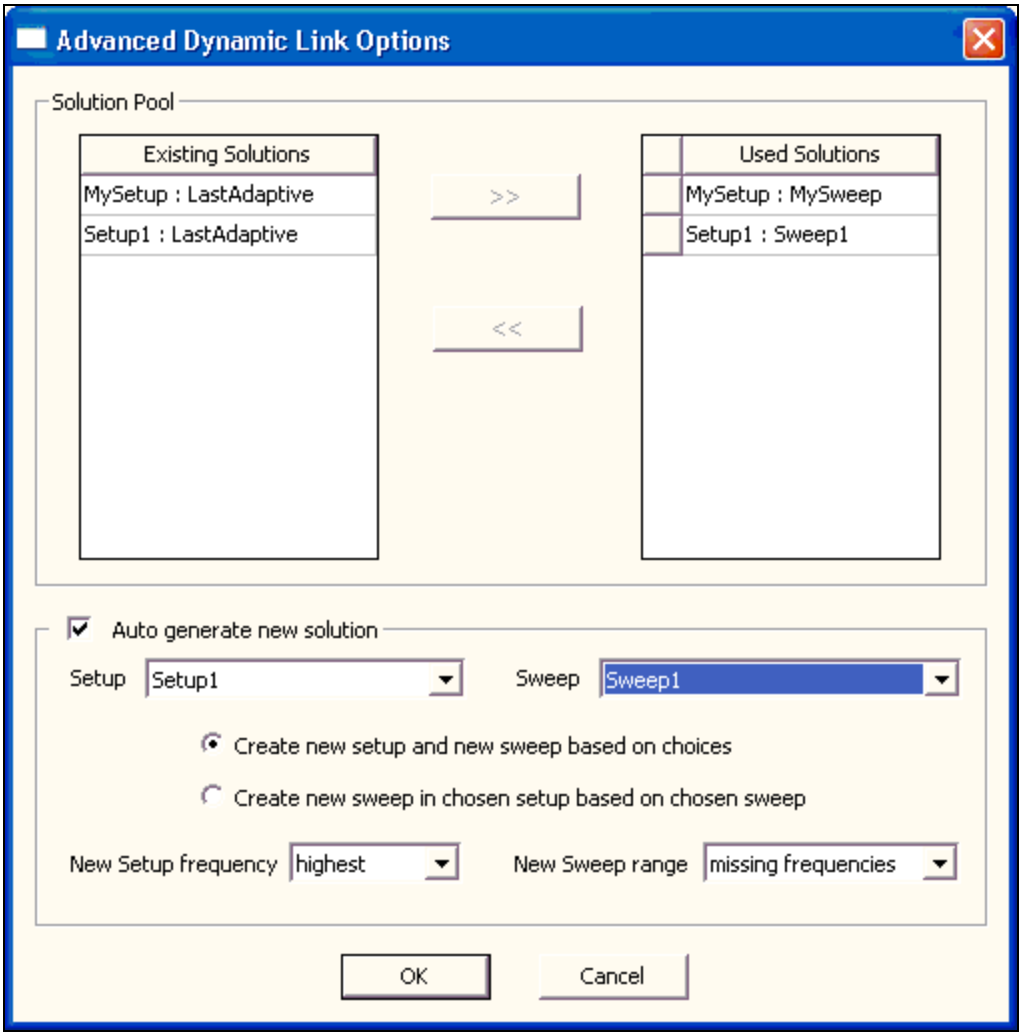
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Design name** specifies the top-level design in the filed solver project.
- **Solution** specifies the solution to use for the Parametric component.
- Click **Select existing solution** to enable the selection field. Then make a selection on the HFSS solution drop-down menu.
- **Transmission line model** specifies that the model specified in the drop-down menu at right are used for transmission lines (HFSS import only)
- Click **Advanced** to specify a custom setup using **Advanced Options**. When **Advanced** setup has been selected, **Advanced Options** becomes active. Click on **Advanced**

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

Options to open the **Advanced Dynamic Link Options** window:



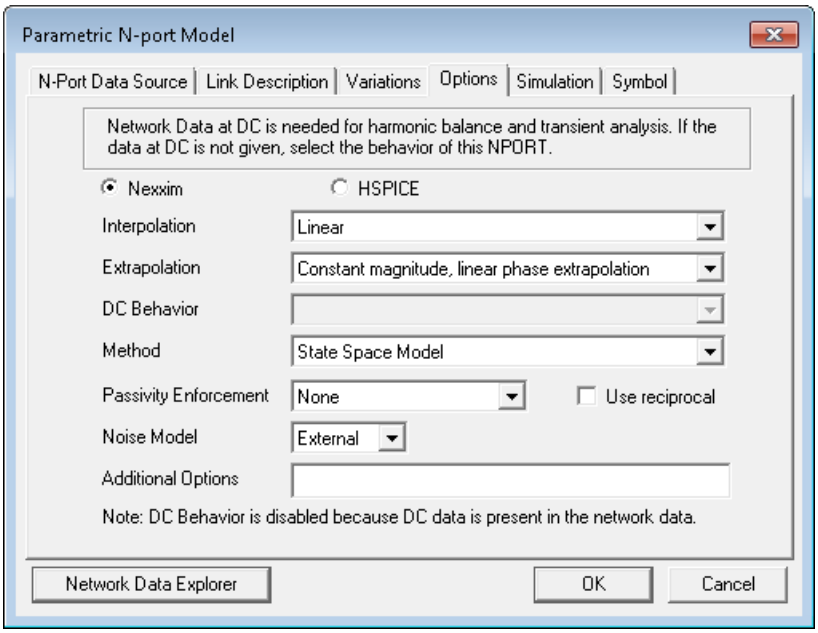
- Select one or more of the **Existing Solutions** to use in the simulation.
- Click **Auto generate new solution** to calculate missing frequency ranges.
- Select the appropriate **Setup** and **Sweep**.
- Use **Create new setup** to specify that a new setup specification are defined. This generates a new adaptive mesh for the analysis. Next, click the **New Setup frequency** field to select the new adaptive frequency for mesh generation: “highest” or “middle”.
- Use **Create new sweep** to specify that a new sweep specification are defined using an existing setup. This uses the existing adaptive mesh for the analysis, which saves simulation time. Next, select a **New Sweep range**: select “all frequencies” to simulate the entire range of frequencies selected, or select “missing frequencies” to simulate only the frequencies not listed in the **Used Solutions** area.

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 **Advanced Dynamic Link Options** window. When the **Link Description** has been completed, click **OK** or select another tab.

Parametric Options Tab

The **Options** tab is used to select the behavior of the selected NPort.



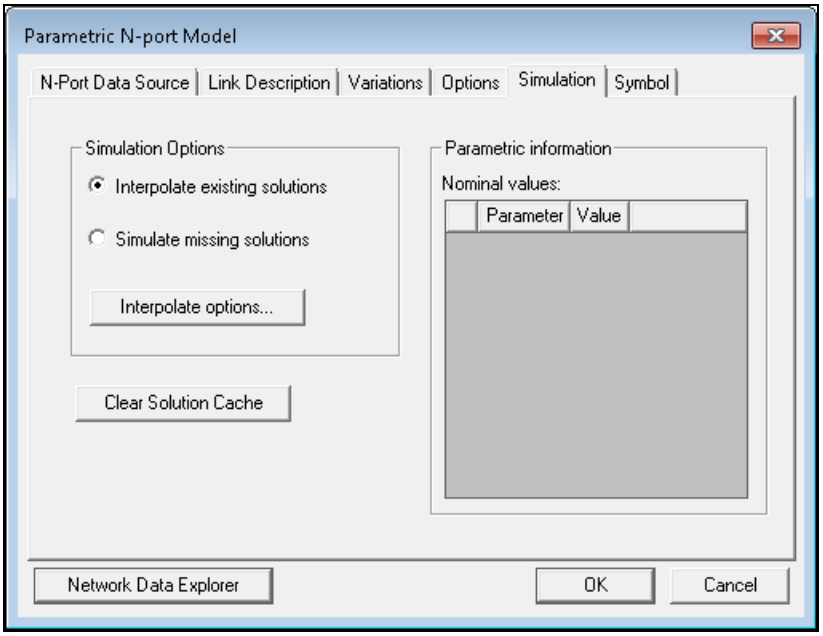
- **Interpolation** — Specifies the interpolation as **Step** or **Linear**
- **Extrapolation** – Specifies the type of extrapolation to use with the NPort
- **DC Behavior** — Specifies the type of DC behavior to use with the NPort
- **Method** – Specifies the method used to convert frequency domain data to the time domain
- **Passivity Enforcement** — Enforces a passivity check which tests whether the S-parameter data is passive or not. Passive devices can only dissipate, or temporarily store energy, but never generate it.
- **Use reciprocal** – Computes the inverse, or reciprocal, at each frequency so the N-port can be used for deembedding. See Deembedding S-Matrices.

When the N-port behavior **Options** have been entered, click **OK** or select another tab.

Parametric Simulation Tab

From the **Simulation** tab, choose the simulation options of the N-Port.

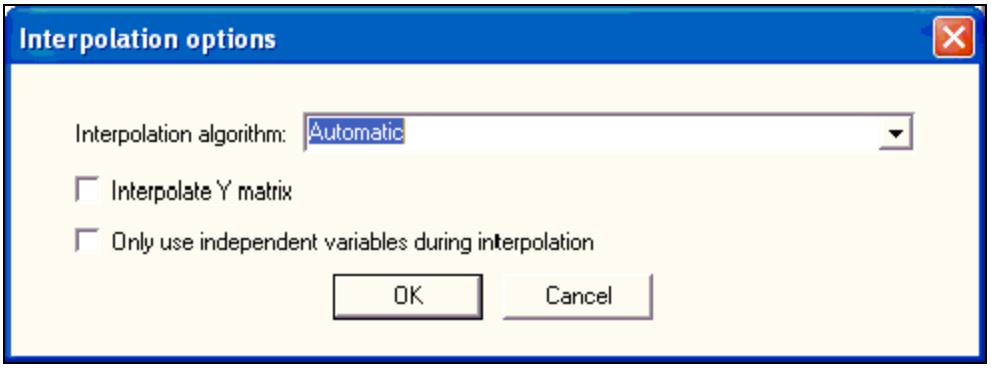
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Interpolate existing solutions** interpolates existing solution data. When **Interpolate existing solutions** is not selected, the solver simulates or calculates any solution data that is needed.
- **Simulate missing solutions** asks HFSS or Q3D to simulate, if the solution is not available.
- **Clear Solution Cache** when clicked clears the solution cache memory.
- **Show available variations** queries HFSS and reports what parameter values have been simulated in HFSS. This information may help you to decide whether to solve additional variations in HFSS or to use interpolation when doing co-simulation with the Circuit solver and HFSS. Though the HFSS project may possess several variables, it may have run simulations only for certain values of those variables. (Each set of variable values defines a *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 **Interpolation options** to open the following window:



- Click the down arrow in the **Interpolation algorithm** field to select an interpolation algorithm option:

  Automatic – Linear interpolation is used if a full grid of solutions is present; otherwise, Inverse Least Squares with Shadowing and Hyperplanes is used.

  Linear – When a full grid of solutions is available, the cube of solutions which surrounds the solution to be interpolated is located. The corners of this cube are linearly averaged to determine the interpolated solution. If a full grid is not available, an error is reported and the interpolation fails.

  Inverse Least Squares with Shadowing and Hyperplanes – Least squares interpolation with shadowing and hyperplanes.

  Inverse Least Squares with Shadowing, no Hyperplanes – Least squares interpolation with shadowing, but no hyperplanes.

- **Interpolate Y matrix** uses the Y matrix as the basis for interpolation. When **Interpolate Y matrix** is not selected, the S matrix is used as the basis. **Interpolate Y matrix** is selected by default. Circuit particulars determine which basis (S or Y) yields better results, and it is not possible to decide beforehand which one works best.
- **Only use independent variables during interpolation** suppresses the calculation of dependent variable values during interpolation.

After choosing interpolation settings, click **OK** to close the **Interpolation options** window.
When the N-port Simulation options have been entered, click OK or select another 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:

The main difference between **Linear** interpolation and **Inverse Least Squares with Shadowing** (ILSS) is as follows:

- **Linear** works only with a full grid of solutions — ILSS works with arbitrary data.
- **Linear** considers only nearest data — ILSS considers data that could possibly be distant, although shadowing mitigates this effect.
- **Linear** is not first-order continuous — ILSS is first-order continuous.

Editing Parametric N-Port Symbol Bus Pins

To edit the pins on the symbol for the N-Port component, right-click the component and select **Edit Symbol Bus Pins**.

The procedure is identical for all the flavors of N-ports. See Editing All N-Port Symbol Bus Pins for details.

Touchstone Calibration Wizard

The Ansys Calibration Wizard is used to characterize the effect of a probe or fixture, and removes the effect from a device under test, by reading and writing Touchstone files. There are three methods of calibration: Two-line, Thru-reflect-line, and Short-open-load-thru. Please note that for measured data, it is often desirable to smooth the data before calibrating the fixtures.

Smoothing can be accomplished using the Network Data Explorer.

Two-Line Method

To implement the Two-Line Method, select **Tools > Calibration Wizard** to open the wizard window. Then select “Two-line method” on the **Type** drop-down menu. The “two line” calibration method removes the effect of the probe fixtures from a device under test and assumes the effect of fixture 1 is identical to fixture 2.

Note:

The two line method assumes the probe discontinuity can be represented as a simple shunt element. A larger probe will generate proportionally larger errors in calibration. Tolerance is at user discretion.

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 can be only one propagating mode on the transmission line. If there are multiple propogating modes, then calibration may fail.

The required input is:

1. Two Touchstone files (*.s2p), each of a thru-line of a given length plus the probe fixtures. The lines must be of different lengths.
2. The imported Zo uses Touchstone file (*.s2p) of an ideal line with no fixture effects. This Touchstone file may come from measurement or simulation. If no file is specified, the wizard approximates the Zo.
3. A Touchstone file (*.s2p) of the selected device under test with fixture effects.

The output is:

- A Touchstone file of the fixture.
- A Touchstone file of the device under test with effect of the fixtures removed.
- If "Export Zo" is selected, the frequency dependent Zo of the calibration lines are written to the designated file.

Thru-Reflect-Line Method

To implement the Thru-Reflect-Line method (TRL), select **Tools > Calibration Wizard** to open the wizard window. Then select "Thru-Reflect-Line" on the Type drop-down menu. The “Thru-Reflect-Line” calibration method removes the effect of the probe fixtures from a device under test. Probe 1 and probe 2 are not assumed to identical.

The required input is:

1. Two Touchstone files (*.s2p), each of a thru-line of a given length plus the probe fixtures. The lines must be of different lengths.
2. For each port users can specify either a single-ended (*.s1p) or a combined measurement (*.s2p) file. When a *.s2p file is specified, Port 1 uses the S11 values and Port2 uses the S22 values:
   - For a short or matched measurement, the surplus inductance must be specified.
   - For an open circuit reflection, excess capacitance must be specified.
   - If no value is given, an ideal reflect is assumed.
3. The imported Zo uses Touchstone file (*.s2p) of an ideal line with no fixture effects. This Touchstone file may come from measurement or simulation. If no file is specified, the wizard approximates the Zo.
4. A Touchstone file (*.s2p) of the selected device under test with fixture effects.

The output is:

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

- Touchstone files for the fixtures corresponding to Port1 and Port2.
- A Touchstone file of the device under test with effect of the fixtures removed.
- If "Export Zo" is selected, the frequency dependent Zo of the calibration lines are written to the designated file.

Short-Open-Load-Thru Method

To implement the Short-Open-Load-Thru method (SOLT), select **Tools > Calibration Wizard** to open the wizard window. Then select "Short-Open-Load-Thru" on the Type drop-down menu. The "Short-Open-Load-Thru" calibration method removes the effect of the probe discontinuities from a device under test and does not assume the port 1 and port 2 are equal.

The required input is:

1. One Touchstone file (*.s2p) of a thru-line of any length.
2. Three Touchstone files for each port corresponding to each measurement configuration: shorted, open and loaded. Each file can either be a single-ended (*.s1p) or a combined measurement (*.s2p) file. When a *.s2p file is specified, Port 1 uses the S11 values and Port2 uses the S22 values:
- For a short or matched measurement, the surplus inductance needs to be specified.
- For an open circuit reflection, the excess capacitance needs to be specified.
- If no value is given, an ideal reflect is assumed.
3. A Touchstone file (*.s2p) of the selected device under test with fixture effects.

The output is:

- Touchstone files for the fixtures corresponding to Port1 and Port2.
- A Touchstone file of the device under test with effect of the fixtures removed.
- If "Export Zo" is selected, the frequency dependent Zo of the calibration lines are written to the designated file.

Paths for File Selection

For most models, the **File Open** window is the initial screen. This topic provides details of file pathnames common to all model imports.

The **PersonalLib** (personal library) folder is located at **<Project Directory>/PersonalLib** ; the **userlib** (user library) folder is located at **<Installation Directory>/userlib** ; and the **syslib** (system library) folder is located at **<Installation Directory>/syslib**, where **<Project Directory>** is the location you specified for the Ansys Electronics Desktop (AEDT)™ electronics systems design platform projects, and **<Installation Directory>** is the directory into which AEDT was installed during setup.

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

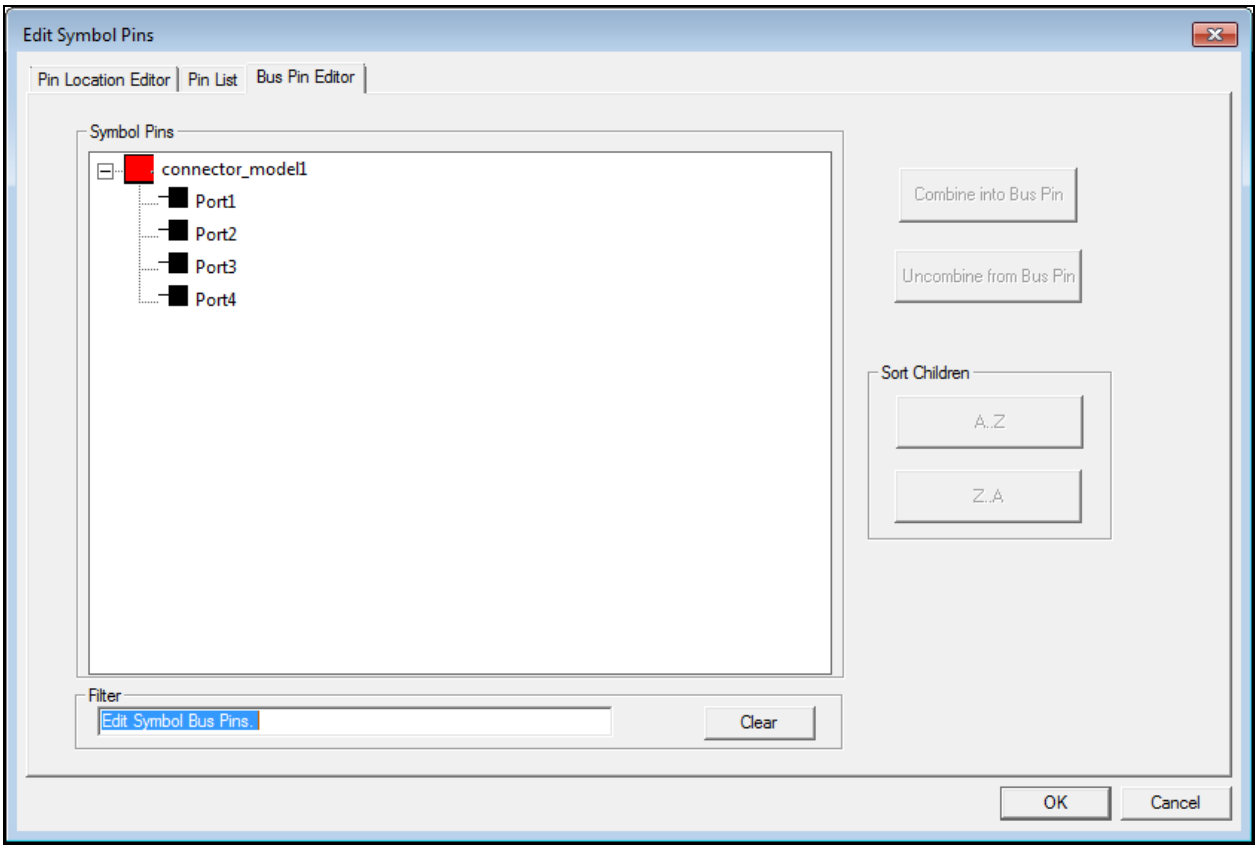
If you select **Use path**, type the name of the file in the **File Name** area, or click the **Look in** field to navigate to the file and record its name. Note that components or libraries imported with Use Path may not be portable when the project is moved to another machine.

When **In project folder** is selected, references to the file in the design are relative to the directory where the project resides. In this case, the path is saved in the project **.aedt** file as a variable (e.g., .lib '$PROJECTDIR/x_113854.lib').

The variable $PROJECTDIR are expanded to the current location of the project when the design is converted to a netlist and run by an analysis tool. Otherwise, an absolute path is saved. If you move a project and its library files together to a new directory, you can preserve the file references by selecting the **Project Folder** option.

Editing All N-Port Symbol Bus Pins

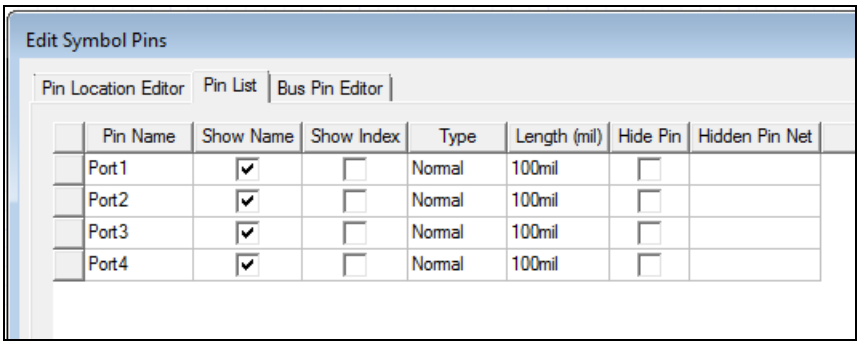
To edit the pins on the symbol for an N-Port, State-Space N-port, or Parametric N-Port component, right-click the component and select **Edit Symbol Bus Pins**. The **Edit Symbol Pins** window opens:



The **N-port data** window reads the file you specify for port information and updates the **Bus Pin Editor** tab display with port names.

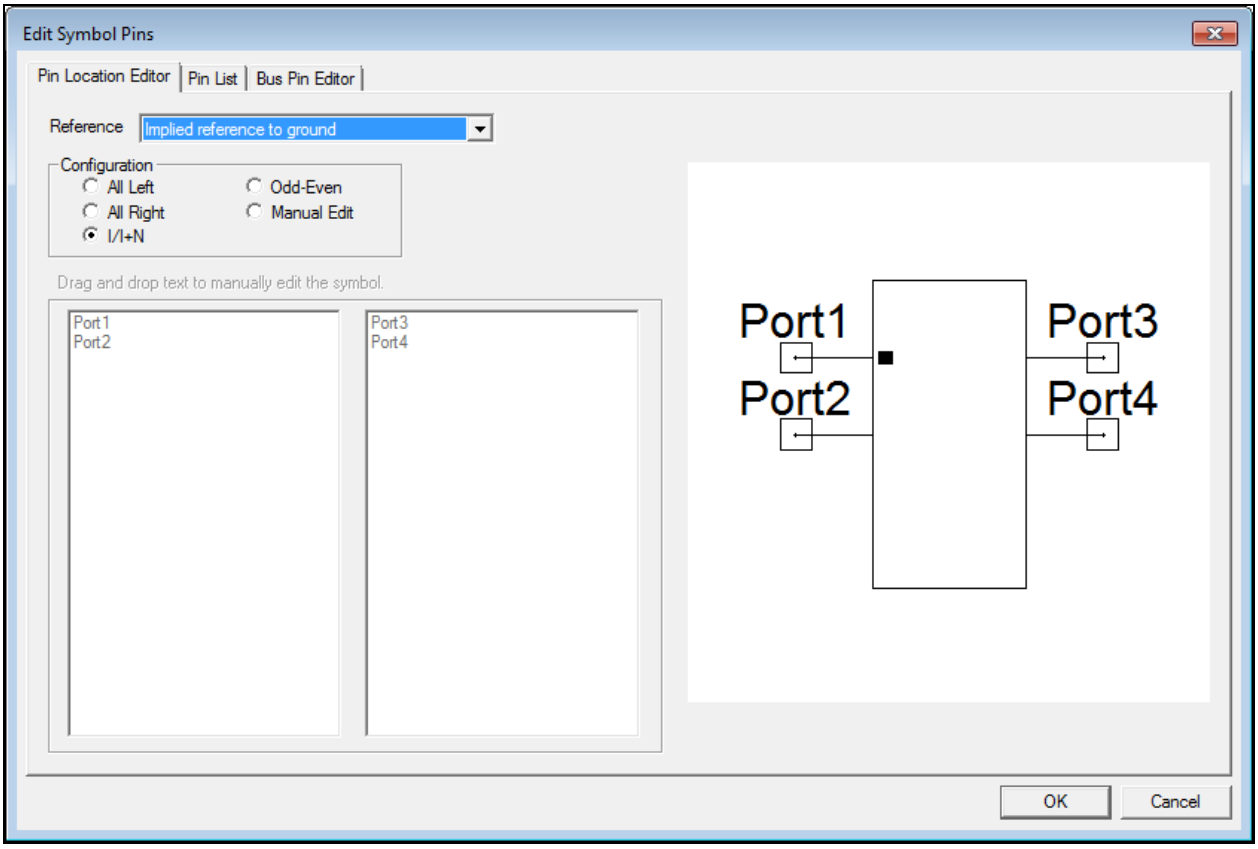
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click Pin List tab:



You can control the display of the pin information (name, index, hidden) with the check boxes.

Click **Pin Location Editor** tab:



You can use the following controls to position pins on the symbol:

- **Reference:** This drop-down menu allows you to select on the following reference node options:

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

— Implied reference to ground

— Show common reference port

— Add individual hidden reference pin per port

— Add individual reference pin per port

> **Note:**
>
> (See “Reference Nodes on S-Parameter Elements” in the S-parameter Technical Notes topic for details on how the Circuit solver handles reference node configurations.

- **Configuration:** The pin direction option may be set to one of the following:

  — All Left

  — All Right

  — I/I+N

  — Odd-Even

  — Manual Edit

  If you select Manual Edit, use the windows at lower left to **Drag and drop text to manually edit the symbol**.

Click **OK**.

Verilog Library Support

A Spectre® netlist can contain include-file references to Verilog® model files. Nexxim can access the definitions of any models that are instantiated in the Spectre netlist. Also create schematic components that instantiate Verilog library models.

The Nexxim implementation includes a built-in Verilog compiler. Compiled mode simulation is supported. Each VerilogA cell is compiled into a C++ file and linked into a DLL. This DLL is loaded during simulation. The precompiled cells are used for each subsequent simulation until a recompilation occurs. A cell is recompiled when a it is modified or when different parameter values are passed to it. You can force a recompile with the option **ahdl.compiled_mode.use_precompiled_cells=0**.

Nexxim also performs encryption and auto-decryption of 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.

Note:

The current release of the Ansys Electronics Desktop (AEDT)™ electronics systems design platform supports only the compiled mode of simulation for Verilog models. Interpretive mode simulation is not currently supported.

Verilog is not supported on Ubuntu Linux

Create a Verilog Symbol

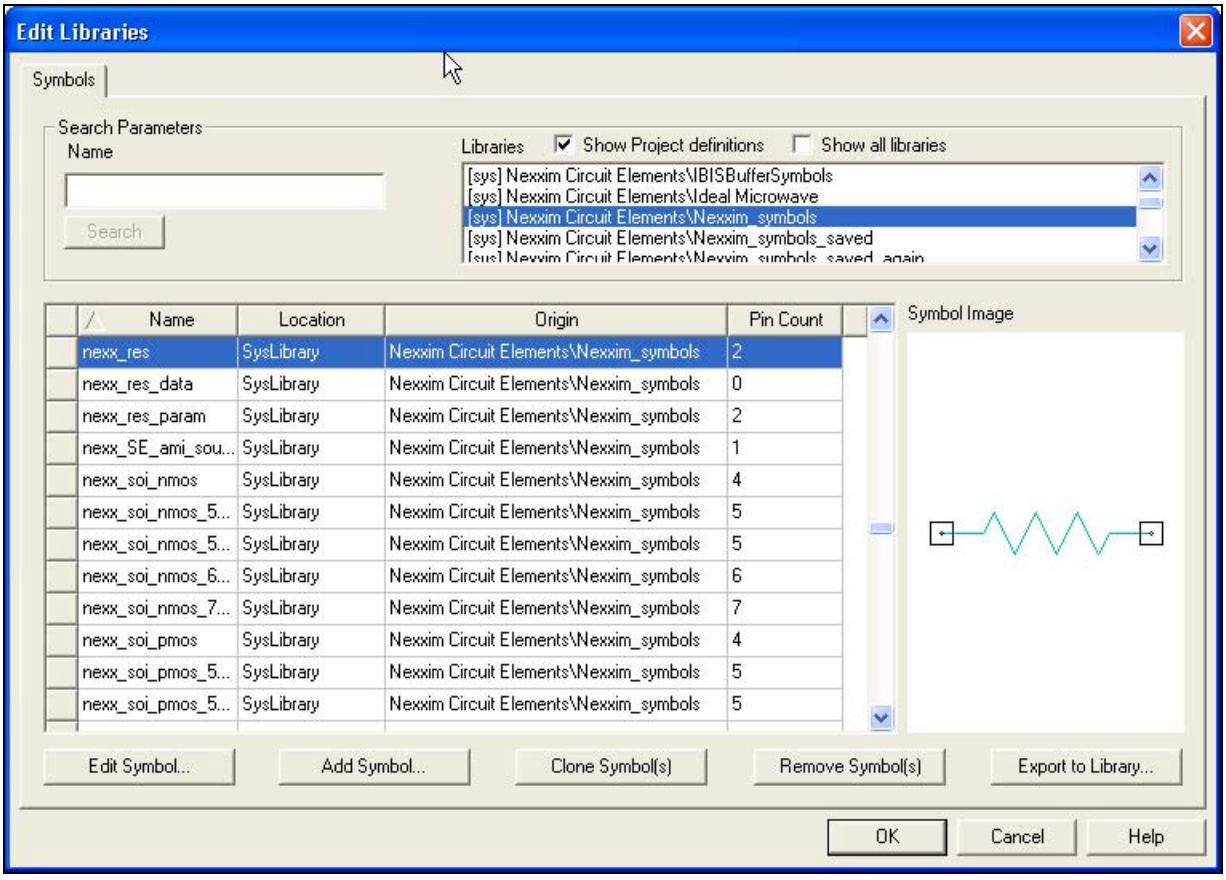
The first step is to create a schematic symbol to represent the Verilog component. A symbol consists of a graphic design with pins for the electrical nodes that connect the component to the circuit. In this example the Verilog symbol is a copy or “clone” of a Nexxim resistor symbol.

(See Using the Symbol Editor in the Component Libraries topic for additional details on creating a new symbol.)

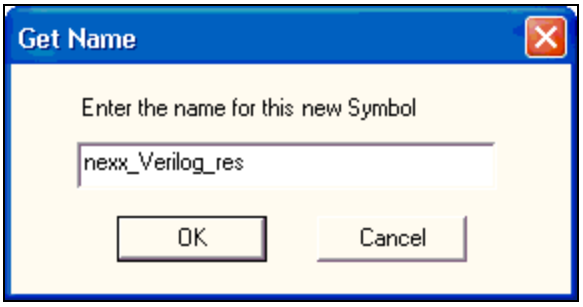
1. Start the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. Insert a new **Circuit** design. Select **None** in the Choose Layout Technology window.
2. From the menu, select **Tools**>**Edit Libraries**>**Symbols**. The **Edit Libraries** window opens with the **Symbols** tab. Ensure both the **Show Project definitions** is checked (on)

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 **Show all libraries** boxes are not checked.



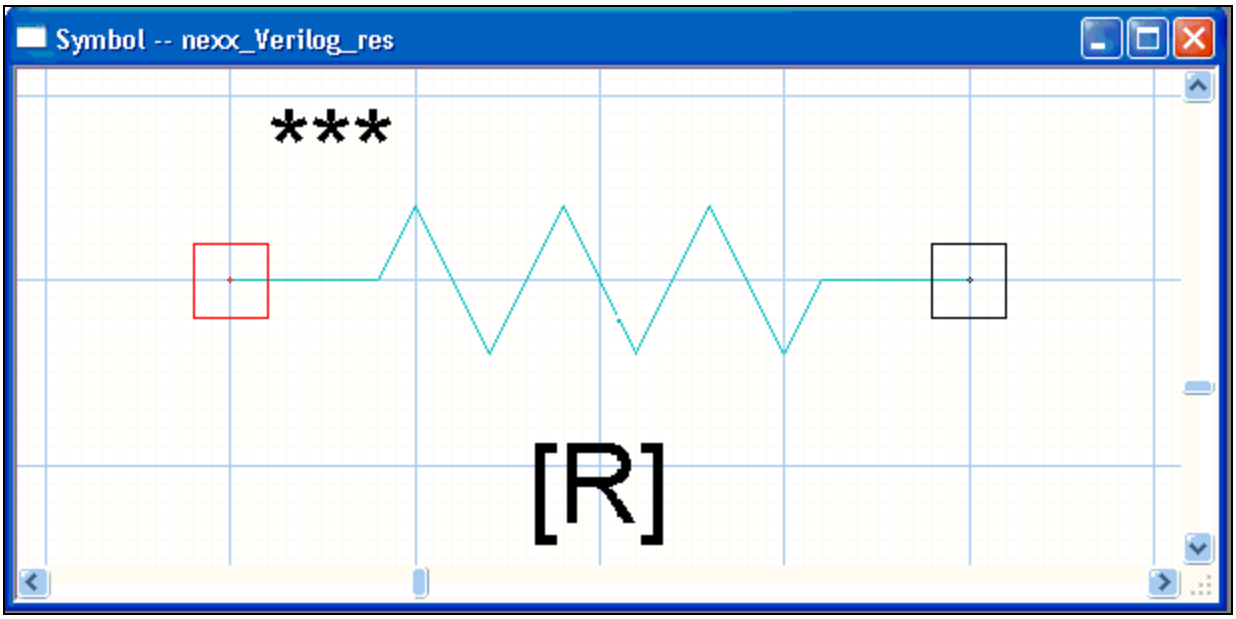
3. Scroll to the symbol library file containing the symbol to clone. In our example, the file is **Nexxim Circuit Elements/Nexxim_Symbols**. Scroll to the symbol, **nexx_res** in our example. Click on that symbol to select it, and click **Clone Symbol** to open the **Get Name** window. Enter the name for the Verilog symbol, in this example **nexx_Verilog_res**.



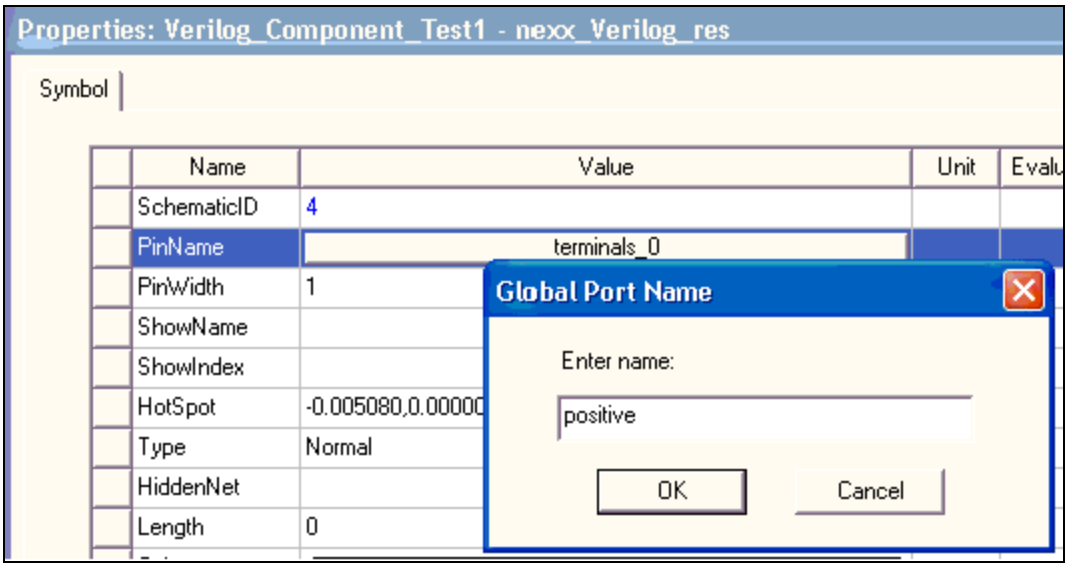
5. Click **OK** to close the **Get Name** window. The new symbol appears in the Edit Libraries window listing as a Project symbol.

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 new symbol and click **Edit Symbol** to open the **Symbol Editor** window with the resistor symbol displayed.
7. Click left terminal so the properties of the terminal appear in the **Properties** window.



8. Click **Value** for the **PinName** property to open the **Global Port Name** window. Rename the terminal to match the node name in the Verilog model. In this example, the terminal has been renamed to **positive**.



9. Select the other terminal and rename it. In our example the new name is **negative**.

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

10. From the menu, select **Symbol>Update Project**. Close the **Symbol Editor** window.
11. From the menu, select **Tools>Edit Libraries>Symbols**. The **Edit Libraries** window opens with the Symbols tab. Ensure the **Show Project definitions** is checked (on) and the **Show all libraries** boxes are unchecked (off).
12. Scroll to the symbol you just created, in our example, **nexx_Verilog_res** in the Project. Select the symbol and click **Export Symbol** to open the Export to user library window.
13. AEDT saves user-defined objects in a **PersonalLib** directory that is specified at install time. To change the location of the **PersonalLib** directory, select **Tools>Options>General Options**.
14. Click the **Create New Folder** icon at the upper-right of the window to create a new folder named **Verilog**. Click new directory to open it, then save the symbol in a new library file named **Verilog_Symbols.aslb**. AEDT prompts you to verify the creation of the new library file.

Now the symbol has been created, the next step is to create a component.

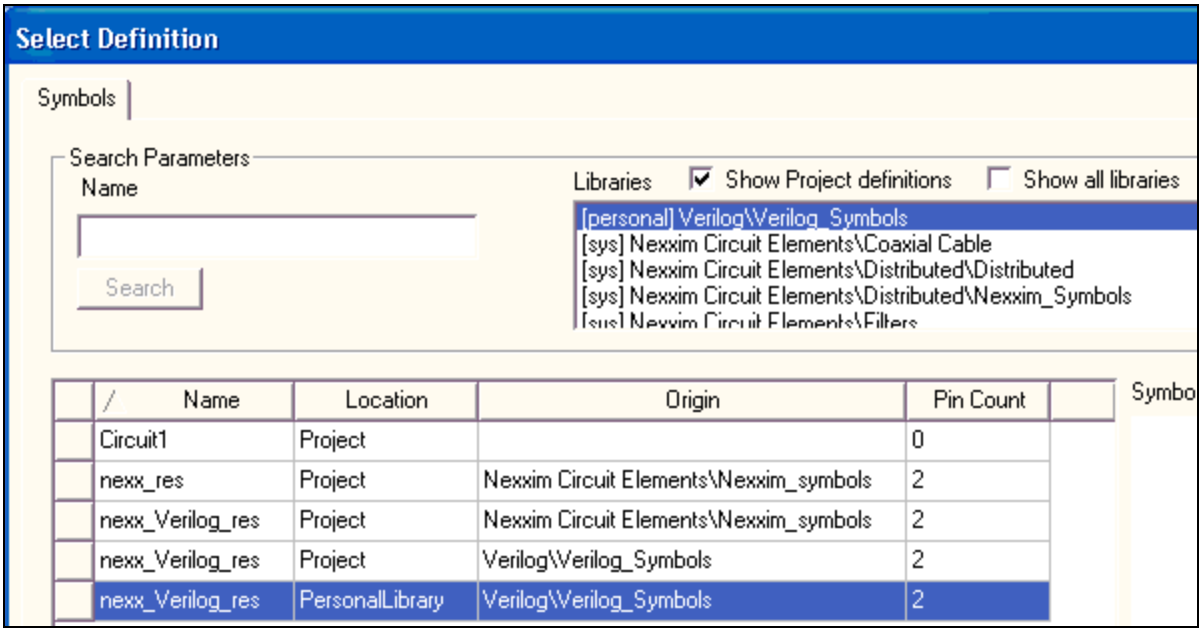
Create a Verilog Component

The component definition provides all the information needed to instantiate the component in the schematic and generate a correct netlist for Nexxim to simulate. In this example the component is copied (“cloned”) from a standard Nexxim resistor.

See Using the Component Editor in the Component Libraries topic for details on creating a new component.

1. From the menu, select **Tools**>**Edit Libraries**>**Components**. The **Edit Libraries** window opens on the **Components** tab. Ensure the **Show Project definitions** is checked (on) and the **Show all libraries** box is unchecked (off).
2. The **Libraries** area at the upper right lists all the categories of Nexxim components. Scroll to the **Nexxim Circuit Elements\Resistors** category, and click on that category. The window reopens to show the Nexxim resistor components. Click on the **RES_** component to select it, and click **Clone Component** to open the **Edit Component** window. Enter the name **RES_Verilog**.
4. From the **Edit Component** window, click the **Symbol [...]** button to open the **Select Definition** window. Ensure the **Show Project definitions** is checked (on) and the **Show all libraries** box is unchecked (off). Select **[personal]Verilog/Verilog_Symbols** on the libraries listing. From the **Personal Library**, select **nexx_Verilog_res**. Then click **OK** to close the **Select Definition** window.

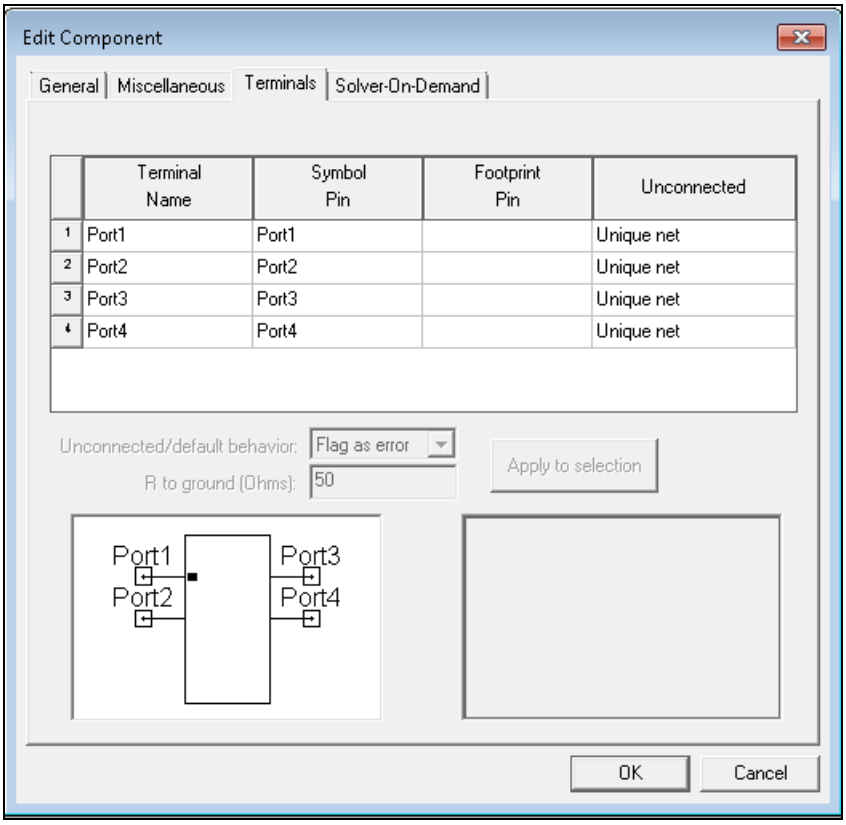
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Select the **Miscellaneous** tab on the **Edit Component** window, and change the **Description** to **Resistor, Verilog**.

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

6. Select the **Terminals** tab and verify the terminal names have been read on the symbol.

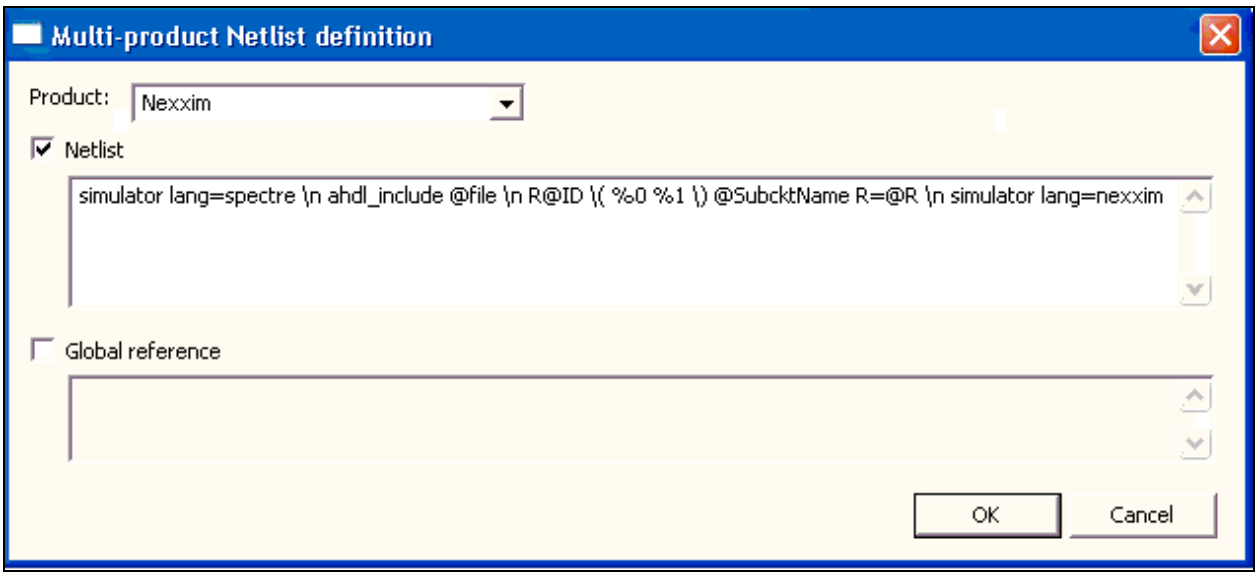


7. Select the **General** tab, and click **Properties** to open the **Edit Properties** window.
8. Select and Remove any properties that should not be used in the Verilog component. In the resistor example, **ADDNOISE** and **DTEMP** are the properties selected for deletion.
9. The component requires a **file** property to link to the Verilog model. Click **Add** to open the **Add Property** window. Enter **file** as the **Name**, click **FileName** for the property type, and enter **required** as the value. When the component is instantiated in the schematic, users set the **file** property by selecting the Verilog library file from a **Select File** window.
10. The component requires a unique name. Click **Add** again. Enter **SubcktName** as the **Name**, click **Text**, and enter **required** as the value. When the component is instantiated, users set the **SubcktName** property to a unique name.

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 general, the name given for the **SubcktName** property must be the same as the module name specified in the Verilog code, or the compilation process fails. If the **SubcktName** is specified as “resistor,” as in this example, the simulation runs successfully irrespective of the name of the Verilog-A module defined in the .va file. The statement “resistor” calls the default resistor model available in Spectre.

11. Add any additional parameters required as input variables in the Verilog model you are accessing. The resistor model in our example uses only variable **R** for the resistance. This property is already in the Nexxim resistor component. When the component is instantiated in the schematic, users set these parameters to the selected values. The instance values are passed to the Verilog model during simulation of the circuit.
12. Click **CosimDefinition** property **Edit** button (or click **Solver-on-Demand** tab of the **Edit Component** window, then click **Edit** for the **Default Netlist** entry). Edit the netlist string to the following:



The **Multi-product Netlist definition** specifies the format for the netlist entry that are generated by instances of the Verilog component. In this string:

- **simulator lang=spectre** turns on the Spectre interpretation of the entries. The equals sign may look like a hyphen.
- **[...]** entries are newlines.
- **@file** is the user-supplied name of the Verilog library file.

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

- **R@ID** is the reference designator, the letter **R** followed by a unique number (e.g., R23).
- **\(** and **\)** are literal left and right parentheses.
- **%0** and **%1** are filled in with the names of the circuit nodes attached to the resistor terminals. If your component has more than two terminals, add **%2**, **%3**, etc.)

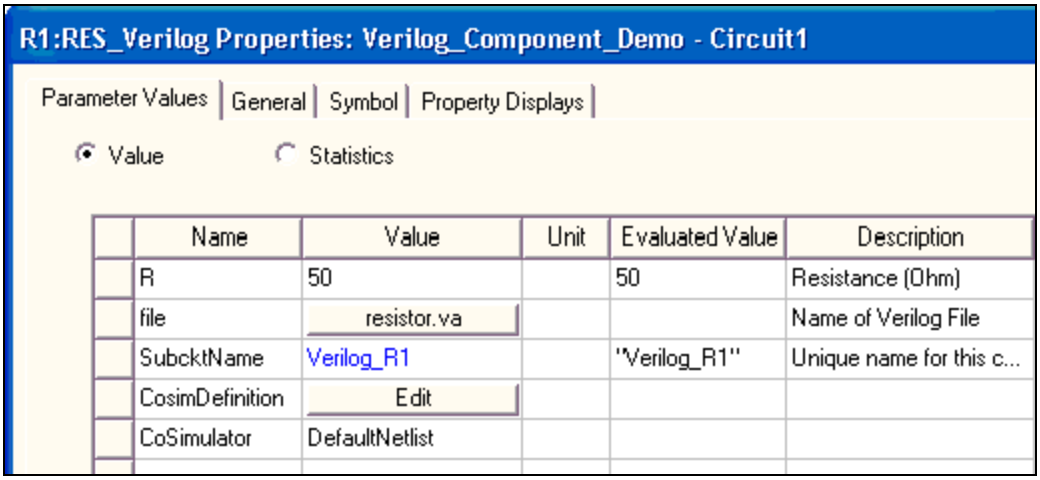
> **Note:**
>
> The spaces around the %0 and %1 separating them on the parentheses are REQUIRED.

- **@SubcktName** is the user-supplied name of the Verilog subcircuit.
- **R=@R** netlists the resistance value, the characters **R=** followed by the value of the **R** property (**@***property* puts the value of *property* in the netlist). Any other properties in the model should be added to the netlist string using this format.
- **simulator lang=nexxim** turns on the Nexxim interpretation of the resulting netlist. The equals sign may look like a hyphen.

13. Click **OK** to close the **Multi-product Netlist definition** window. Click **OK** to close the **Properties** window. Click **OK** to close the **Edit Component** window.
14. Select the new component in the **Edit Libraries** window and click **Export to Library**. Open the **Verilog** directory under **PersonalLib**, and save the resistor component in a new library file, **Verilog_Resistors.aclb**. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform prompts you to confirm the creation of the new file.
15. This completes the process for creating the Verilog component.
16. View the **Component Libraries** window, and click the **Symbols** tab. Click **Proj** to display the Project components. The new component appears in the window.
17. To create an instance of the component, double-click the **RES_Verilog** entry, drag the cursor into the design area, and select **Place and Finish** on 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.

18. Select the resistor and open the **Properties** window:



19. Set the **R**, **file**, and **SubcktName** properties. Click **OK** to close the Properties window.
20. Here is a listing of file **resistor.va**:

```
// Simple resistor
`include "disciplines.vams"
module Verilog_R1(p,n);
input p,n ;
parameter real R=1.0 from (0:inf);
electrical p,n;
real ir ;
analog begin
ir = V(p,n)/R;
I(p,n) <+ ir ;
end
endmodule
```

21. Connect the component and simulate.

Supported Verilog Features

The following features of Verilog-A are supported in Nexxim (not supported on Ubuntu Linux).

1. Operators: all except replication operator '{{}}'
2. All built-in mathematical functions
    a. Basic functions—pow, sqrt, etc.
    b. Trigonometric functions
    c. Hyperbolic and inverse hyperbolic functions

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

3. Natures: ‘voltage’
4. Disciplines: ‘voltage’, ‘electrical’
5. Current/Voltage access functions
6. Analog nodes and branches
7. Parameters
    a. ‘real’ and ‘integer’
    b. Scalar parameters
    c. Vector parameters with constant array initializations
    d. Value range specification —‘from’ and ‘exclude’, use of ‘inf’
8. Variables
    a. ‘real’ and ‘integer’
    b. scalar and vector
9. Genvars
10. Vectors
    a. Ports declared as vectors
    b. Vector variables
    c. Vector nodes
    d. Vector branches
    e. Bit-selects of vector variable/node/branch/parameter

- Index of the bit is a loop or generated index-variable
- Index of the bit is a parameter

11. Analog contributions, to assign currents or voltages
    a. A branch with a voltage contribution in one evaluation and a current contribution in another.
12. Analog sequential blocks containing analog statements
13. Behavioral statements
    a. Procedural assignment
    b. Control statements (‘if-else’, ‘case’)
14. Instantiations
    a. VerilogA instantiations
    b. Spectre device instantiations
15. User-defined functions
16. Analog operators
    a. Time derivative (‘ddt’)
    b. Derivative operator (‘ddx’)
    c. ‘transition’ operator
    d. ‘last_crossing’ function

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

e. Laplace filters (laplace_zp, laplace_nd)
f. Limited exponentiation (‘limexp’)
g. Absolute delay operator (‘absdelay’)
h. Slew filter (‘slew’)

17. Analog events
    a. Global events

- ‘initial_step’, ‘final_step’
- ‘timer’, ‘cross’

18. Noise analysis
    a. white noise
    b. flicker noise
19. Analysis-dependent functionality using ‘analysis’ function
    a. AC stimulus (‘ac_stim’)
20. Compiler directives
    a. 'include directive
    b. 'define directive to define constants
21. System functions
    a. $temperature
    b. $vt
    c. $abstime
    d. $realtime
    e. Table based interpolation and lookup function—$table_model (interpolation data either in data file or in VerilogA file).
22. System tasks
    a. Simulation control ($finish, $stop)
    b. Analog kernel control ($bound, $step, $discontinuity)
    c. Display tasks ($display, $write, $strobe, $monitor, $debug)
    d. $table_model

Supported keywords, functions, and system tasks

$abstime	asinh	if
$bound_step	atan	inf
$debug	atan2	initial_step
$discontinuity	atanh	inout
$display	begin	input
$finish	branch	integer

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

$function	case	laplace_nd
$monitor	ceil	laplace_zp
$realtime	cos	last_ crossing
$stop	cosh	limexp
$strobe	cross	ln
$table_model	ddt	log
$temperature	ddx	max
$vt	default	min
$write	else	module
‘define	end	or
‘else	endcase	output
‘elseif	endfunction	parameter
‘endif	endmodule	pow
‘ifdef	exclude	real
‘ifndef	exp	sin
‘include	final_step	sinh
‘undef	flicker_ noise	slew
abs	floor	sqrt
absdelay		tan
ac_stim	from	tanh
acos	function	timer
acosh	genvar	transition
analog	hypot	white_ noise
analysis	generate	
asin	ground	

Keywords supported with workarounds

Parsing for these keywords is skipped.

abstol	discrete	nature
access	domain	idt_ nature
continuous	enddiscipline	potential
ddt_nature	endnature	units
discipline	flow	

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

Spectre Compatibility

Nexxim VerilogA is compatible with Spectre.

The directive **ahdl_include** for specifying the hdl file is supported.

HSPICE Compatibility

Nexxim VerilogA is compatible with HSPICE.

The directive **ahdl_include** for specifying the hdl file is supported.

The option **hdlpath** for specifying the location of the hdl files is supported.

The environment variable **HSP_HDL_PATH** for specifying the location of the hdl files is supported.

Hdl files can be specified with or without the extension.

Parameter names to Verilog instances are case-insensitive.

NOTE: For, while, and repeat loops are NOT supported.

CPM Model Support

The **Pin Connectivity** window allows you to automatically connect CPM (Chip Power Model) components to existing components. Follow these steps to use this feature.

1. Load or generate a model containing a component with pins that can be connected to a CPM component.For fastest pin-matching, ensure the model component's pins possess names that correspond to the pins specified by the CPM component. If the pin names do not match, the Automated CPM Chip Connection attempts to match pins geometrically. When these models are created, they behave like SPICE models and have regular symbols in the schematic, but the corresponding footprints in the layout show the positions of the pins specified in the CPP (Chip Package Placement) header.
2. To place a CPM component, open the **Component Libraries** window, click the **Symbols** area, and click **Import Models**.
3. From the area, select the **CPM** icon to open the **Import Components** window. After that, follow the procedure for importing Spice components (See SPICE Library Support).

Once the component has been placed in the schematic, its Property Window will show a checkbox property named **NetlistAsSubcircuit**. Enabling the checkbox affects only what is written to the netlist for simulation. When the checkbox is enabled, the Spice model component will be netlisted inside its own subcircuit along with its file reference. This means the model name (taken from the Spice file) cannot conflict with the same model name in a different Spice file netlisted for a different component. If two Spice files with identical model names are read into the

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

global scope, only data from one of the spice files will be used for all components referencing that model name, causing a simulation error. Checking the ”NetlistAsSubcircuit” property will move the file out of the global scope and prevent this problem.

Automated CPM Chip Connection And Placement

CPM (Chip Package Model) components can be created by importing special SPICE files that contain additional information that is encoded using the CPP (Chip Package Protocol).

- The CPP is a set of statements specifying the location, the net information, and the SPICE node name for each package pin (the landing pad of flip-chip bumps or the chip landing pad of bonding-wires).
- The CPP is embedded within a SPICE compatible file header as comments.

When these models are created, they behave like SPICE models and have regular symbols in the schematic, but the corresponding footprints in the layout show the positions of the pins specified in the CPP header.

To place a CPM component, open the **Component Libraries** window, click the **Symbols** area, and click **Import Models**. From the area, select the **CPM** icon to open the **Import Components** window. After that, follow the procedure for importing Spice components (See SPICE Library Support).

Exporting Network Design Data

Several tools generate frequency-dependent network design data as the result of the analysis. The results can be saved (exported) to files for use by other tools.

Exporting LNA Network Data

You can export the frequency-dependent network data that results from a linear network analysis to a Touchstone™ file for use by other analyses. Results can be exported from a design created with either the **Schematic Editor** or the Netlist Editor.

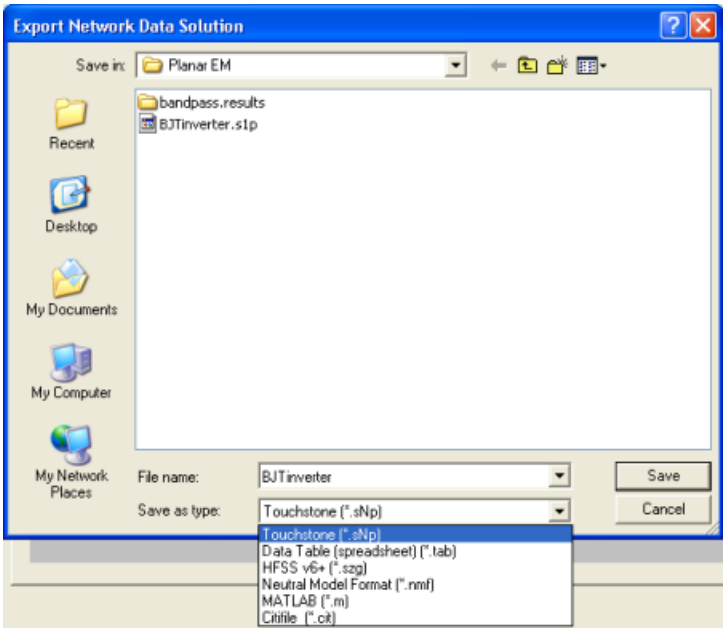
1. Define and run the Linear Network Analysis.
2. In a schematic design, expand the icon for the design. Expand the **Analysis** entry, right-click the LNA solution setup and select **Export Solution**.

   In a netlist design, right-click the icon for the netlist design and select **Export Solution** to open the **Solutions** window.
3. From a schematic, ensure the selected setup name and simulation name are shown in the listings in the **Simulation** fields. (With a netlist design, these fields are read-only.)
4. Select the type of data to save (S, Y, Z, Zo, or Gamma). The choices depend on the results that are available.
5. Select the frequencies for which data can 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.

- To save the data for all frequencies, check **Display All Freqs**.
- To save the data for selected frequencies, leave **Display All Freqs** unchecked and make a selection on the frequencies drop-down menu.

6. Optionally, select **Check Passivity**. EM Design reports any passivity violations in the matrix data. Optionally, set the **Passivity Tolerance** (default 0.0001 or .01%). (See the S-Parameter Technical Notes for details.
7. Select the results and frequencies to be exported, then click **Export**. The **Export Network Data Solution** window opens:



Select a directory for the export in the **Save in:** field. Select a file name for the exported data in the **File name:** field. Select a file type on the **Save as type:** list, , then click **Save**.

Options for file format are:

- Data Table (spreadsheet) (**.txt** file extension)
- Touchstone (**.sNp** file extension)
- Neutral Model Format (NMF) (**.nmf** file extension)
- MATLAB (**.m** file extension)
- CITIfile (**.cit** file extension)

8. The **Specify Export Reference Impedance** window opens to specify the reference impedance for the Touchstone data:

- Select **Override Solution Renormalization** (the default) and set the impedance as selected (50 Ohms is the default). The Touchstone file data are renormalized to the port impedance you specify, if the impedance is not the same as that used for 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.

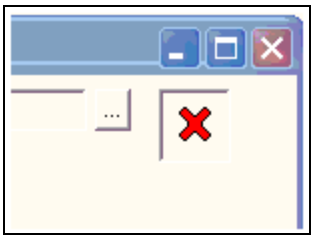
analysis. Selecting **Do Not Override Solution Renormalization** inactivates the Impedance field.

- Choose to Export Noise Data with Touchstone files in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform.
- Choose to include Gamma and impedance comments in the Touchstone file (the default), or to omit them. Omitting the comments can make the Touchstone file much smaller.
- When the impedance information is complete, click **OK**.

9. Click **Close** on the **Solutions** window, or leave the window open to make multiple exports, as described next.

Making Multiple Exports

If you want to export several solutions representing variations on the design, leave the **Solutions** window open. Make the selected changes to the circuit. The box at the upper-right of the **Solutions** window displays a red X to show the solution is invalid:

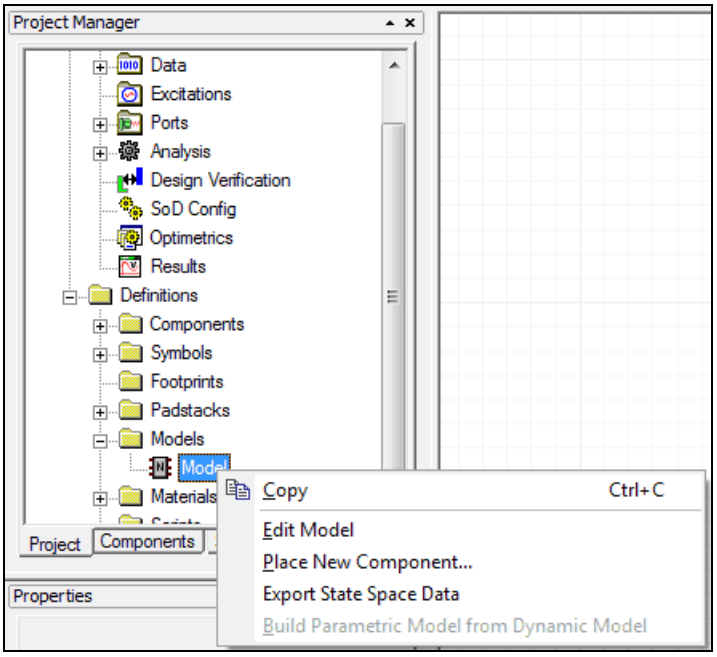


Now, rerun the analysis. The red X disappears to show that a valid solution is available for export. Save the new result as described earlier.

Exporting Network Data to a State Space File

You can export data to a State Space File (*.sss) by right-clicking and selecting **Export State Space Data** under the Definitions/Models folder in the **Project Manager** window. The **Export State Space Data** menu item is enabled only if the model has state-space data available. State space data is available when an SSS file is selected as the data source or when the model has an underlying SSS file associated with it.

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



Choosing **Export State Space Data** opens a **Save File As** window. When you provide a file name and location, the .sss file is generated and updated with the port names in the model.

Importing and Exporting Netlist Data

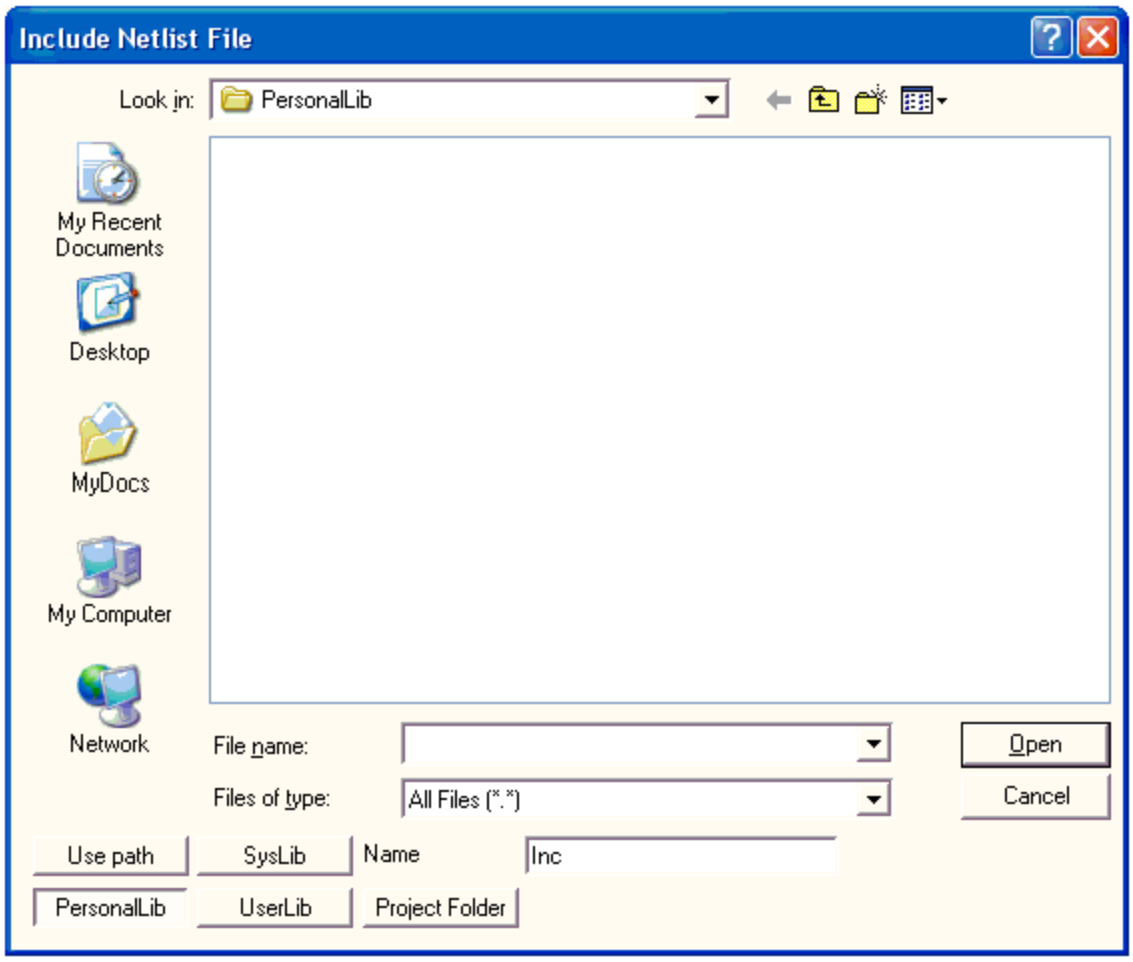
A Circuit schematic project can import netlist design information from netlist files, include files, and library files. A netlist representing a Circuit schematic or netlist project can be exported for future use.

This feature is deprecated but can be used to edit components with netlist models in legacy projects. Netlist models are renamed to Circuit models.

Accessing Netlist Files

A netlist file contains one or more netlist statements. To add a netlist file to a schematic, expand the icon for the design in the **Schematic Editor** by clicking on the plus sign (+) button (the design typically has a name such as “Circuit1”). Right-click the **Data** icon and select **Add Netlist File**. The **Include Netlist File** 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.



Select one of the standard folders if applicable, select the file, then click **OK**.

> **Note:**
>
> The **syslib** directory is reserved for libraries supplied with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform

If the netlist file is not in one of the standard folders, select **Use path**, enter the complete path and file name. Use the **Look in** field to browse for the directory. When the path and file name are correct, click **OK**. (Note that components or libraries that are imported with **Use Path** may not be portable if the project is moved to another machine.)

When **In project folder** is selected, references to the file in the design are relative to the directory where the project resides. In this case, the path is saved in the project **.adsn** file as a variable such as:

```
.lib '$PROJECTDIR/x_113854.lib'
```


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

The variable $PROJECTDIR are expanded to the current location of the project when the design is converted to a netlist and run by an analysis tool. Otherwise, an absolute path is saved. If a project and its library files are moved to a new directory, preserve their file references by selecting the **Project Folder** option.

Include Files in Circuit Netlists

The **.INCLUDE** statement identifies a file to insert in its entirety into the netlist at the point of the **.INCLUDE** statement:

```
.INCLUDE file_reference
```

Include file pathnames, and all pathnames used in the Desktop, must refer to subdirectories using a syntax that is compatible with the platform operating system. Some operating systems use a forward slash (/) as a path separator, while others use a backslash (\). The following example uses a backslash, which is the valid separator used in Windows™ systems. The Desktop ensures the consistent and compatible use of slash designators in netlist file paths for each of the following platforms: Windows and Linux™. For more information see File References.

For example:

```
.INCLUDE 'diode_model_statements'
.INCLUDE 'models\diode_models'
.INCLUDE 'C:\simulator\model_statements\diode_model_statements'
```

The complete contents of the include file are added to the calling netlist. The included file can contain further **.INCLUDE** files to be processed.

> **Note:**
>
> None of the included files can refer to any other file in the sequence of included files. A loop of included files causes an unrecoverable error condition. However, Nexxim cannot detect this error in advance, so no warning message is given.

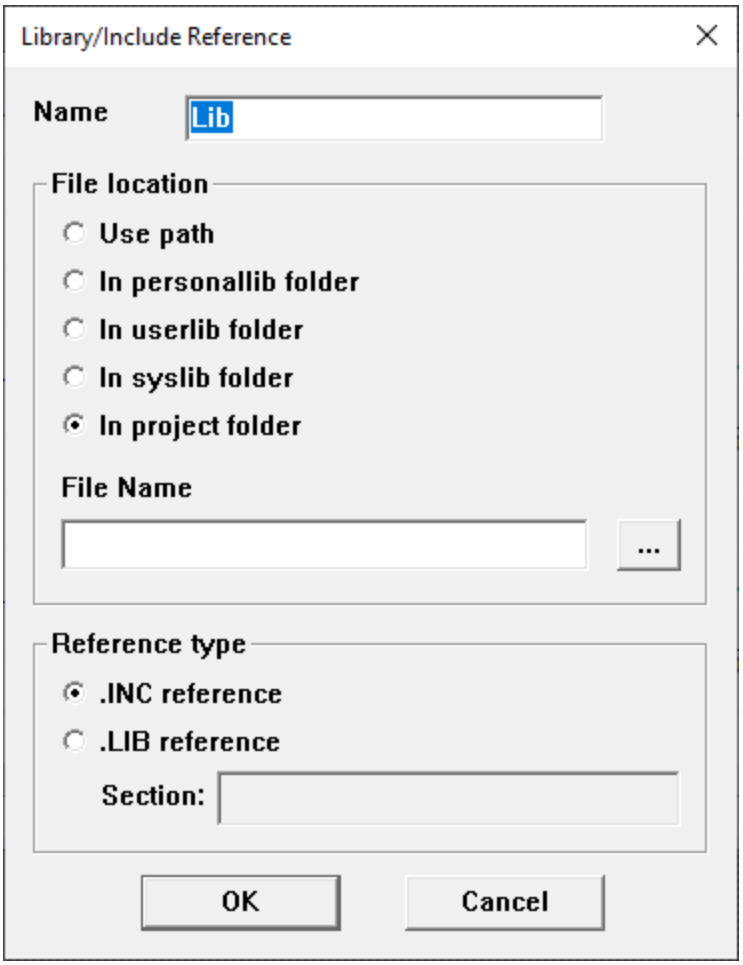
Accessing Include Files

A schematic design can reference an include file (e.g., frequency-dependent model data is commonly written into a separate file rather than entered into a **Property** window). To add an include fileto a design, follow these steps.

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Data** and select **Add Library/Include Reference** to open the

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

Library/Include Reference window.



2. Enter a name for the *.inc* file in the **Name** field.
3. From the **File location** area, choose the folder in which the *.lib* library file is stored (e.g., **In project folder**). If the library file is not in one of the default folders, choose **Use path** and complete steps a-b. Otherwise, skip to step 3.

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

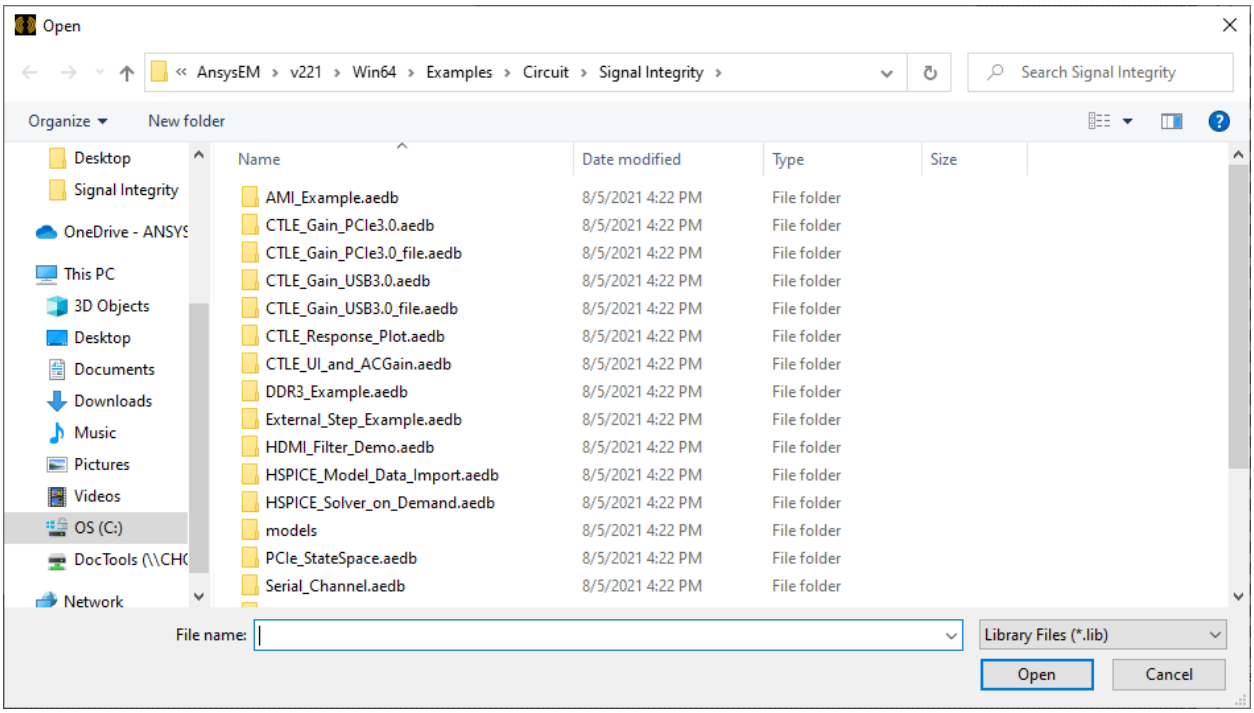
Note:

When **In project folder** is selected, references to the file in the design are relative to the directory where the project is saved (e.g., the path is saved in the project **.adsn** file as a variable: `.inc '$PROJECTDIR/x_113854.inc'`). The variable `$PROJECTDIR` is expanded to the current location of the project when the design is converted to a netlist and run by an analysis tool. Otherwise, an absolute path is saved. If a project and its include files are moved to a new directory together, select **In project folder** to preserve the file references.

Note:

In syslib folder is reserved for libraries supplied with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform.

a. Click the **[...]** button to open an explorer window. Then navigate to and select the appropriate *.inc* include file.



b. Click **Open** to return to the **Library/Include Reference** window. The path to the *.inc* file is displayed in the *File Name* field.

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

> **Note:** Directories accessed with **Use Path** are not exported if the design is moved to another computer, and will be inaccessible.

4. Ensure **.INC reference** is selected in the **Reference type** area (the default). The **Section** field will remain blank/deactivated.
5. Click **OK** to add the library file data to the design.

The procedure is complete.

Library Files in Circuit Netlists

Library files contain one or more blocks of netlist statements. Each block in the library file has the format:

```
.LIBblock_name
lib_block_statements
.ENDL
```

The library file block can contain any valid netlist statements. LIB blocks are processed after all the other statements in the netlist have been processed, and the order of execution of LIB block statements cannot be predetermined.

To read and process in the statements from a library file block, the netlist uses a library call:

```
.LIBfile_referenceblock_name
```

The rules for *file_reference* are the ones given in the module on File References.

A **.LIB** block in a given library file can contain calls to blocks in the same library file or in other library files, but a **.LIB** block may not call a block with its own block name in its own file (that is, a **.LIB** block cannot call itself).

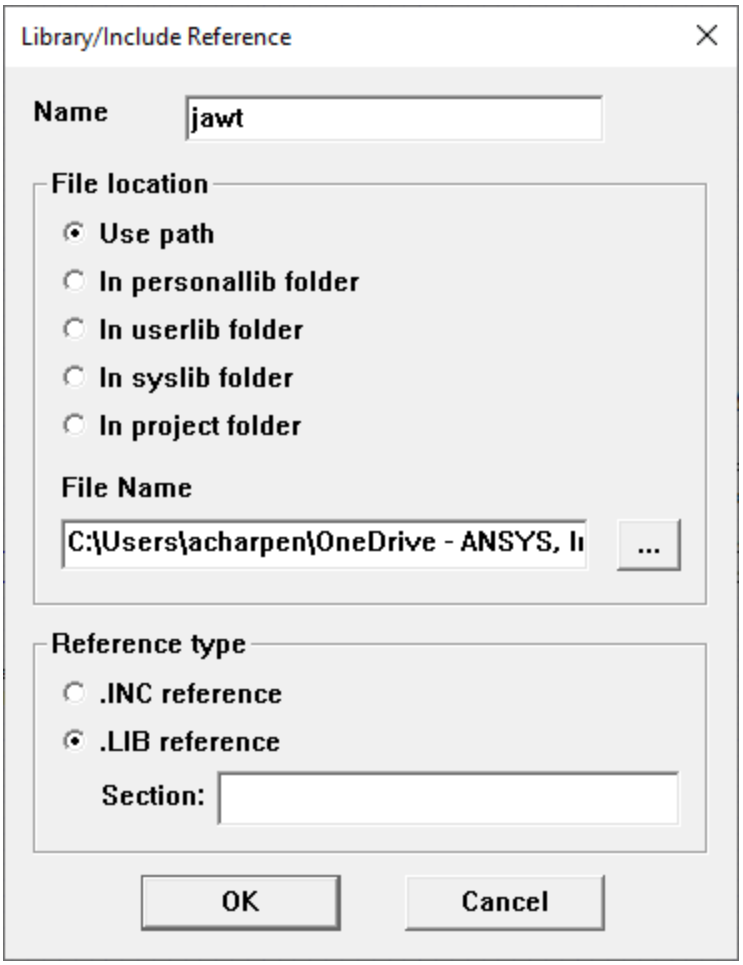
Accessing Library Files

A schematic design can reference a library file block. To add a library file to a design, follow these steps.

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Data** and select **Add Library/Include Reference** to open the

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

Library/Include Reference window.



2. Enter a name for the *.lib* library file in the **Name** field.
3. From the **File location** area, choose the folder in which the *.lib* library file is stored (e.g., **In project folder**). If the library file is not in one of the default folders, choose **Use path** and complete steps a-b. Otherwise, skip to step 3.

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

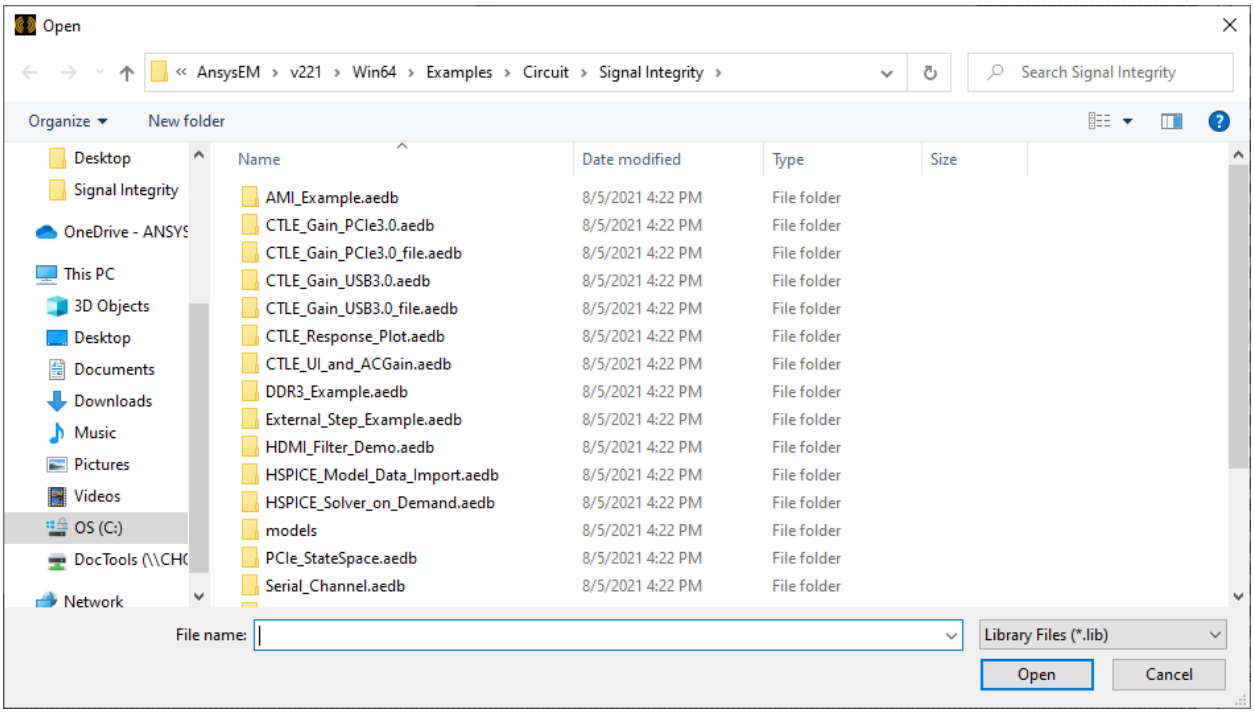
Note:

When **In project folder** is selected, references to the file in the design are relative to the directory where the project is saved (e.g., the path is saved in the project **.adsn** file as a variable: `.lib '$PROJECTDIR/x_113854.lib'`). The variable `$PROJECTDIR` is expanded to the current location of the project when the design is converted to a netlist and run by an analysis tool. Otherwise, an absolute path is saved. If a project and its library files are moved to a new directory together, select **In project folder** to preserve the file references.

Note:

In syslib folder is reserved for libraries supplied with the Ansys Electronics Desktop (AEDT)™ electronics systems design platform.

a. Click the **[...]** button to open an explorer window. Then navigate to and select the appropriate *.lib* library file.



b. Click **Open** to return to the **Library/Include Reference** window. The path to the *.lib* file is displayed in the *File Name* field

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

> **Note:** Directories accessed with **Use Path** are not exported if the design is moved to another computer, and will be inaccessible.

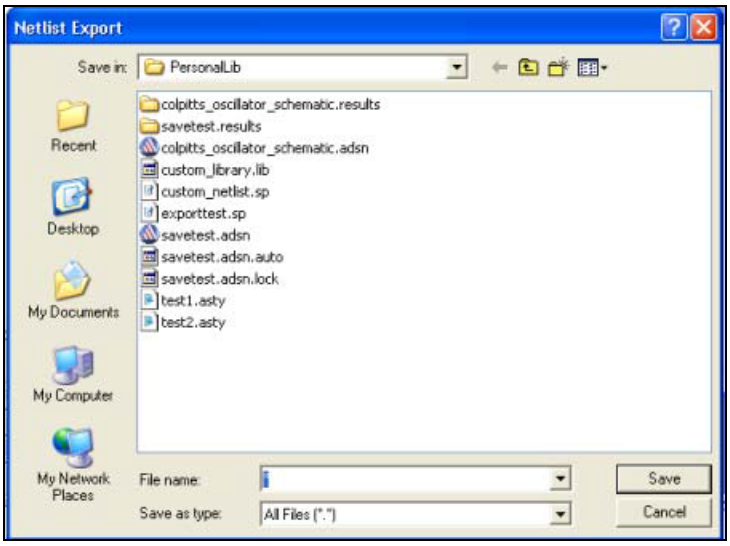
4. Ensure **.LIB reference** is selected in the **Reference type** area (the default) to activate the **Section** field.
5. Enter a name in the **Section** (block) field.
6. Click **OK** to add the library file data to the design.

The procedure is complete.

Exporting Netlist Data

Follow these steps to export the netlist representation of a schematic design to a file.

1. Click **Browse Netlist** to make the **Netlist Editor** the active window.
2. Click on **Netlist** in the top menu bar, and select **Export**. The **Netlist Export** window opens:



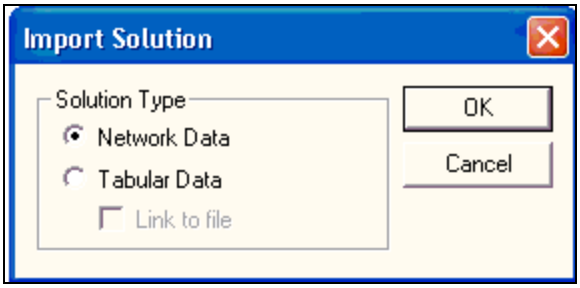
3. From the **Save in** field, browse to an appropriate directory to save the netlist file.
4. From the **File name** field, enter the complete name of the file, including the extension if any.
5. Click **Save** to save the netlist file and close the window, or click **Cancel** to cancel the export operation without saving anything.

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

Importing Solution Data for Postprocessing

All the Ansys Electronics Desktop (AEDT)™ electronics systems design platform simulators allow you to import solution data from other tools for reports and other postprocessing as available. Follow these steps to import solution data into a Circuit design.

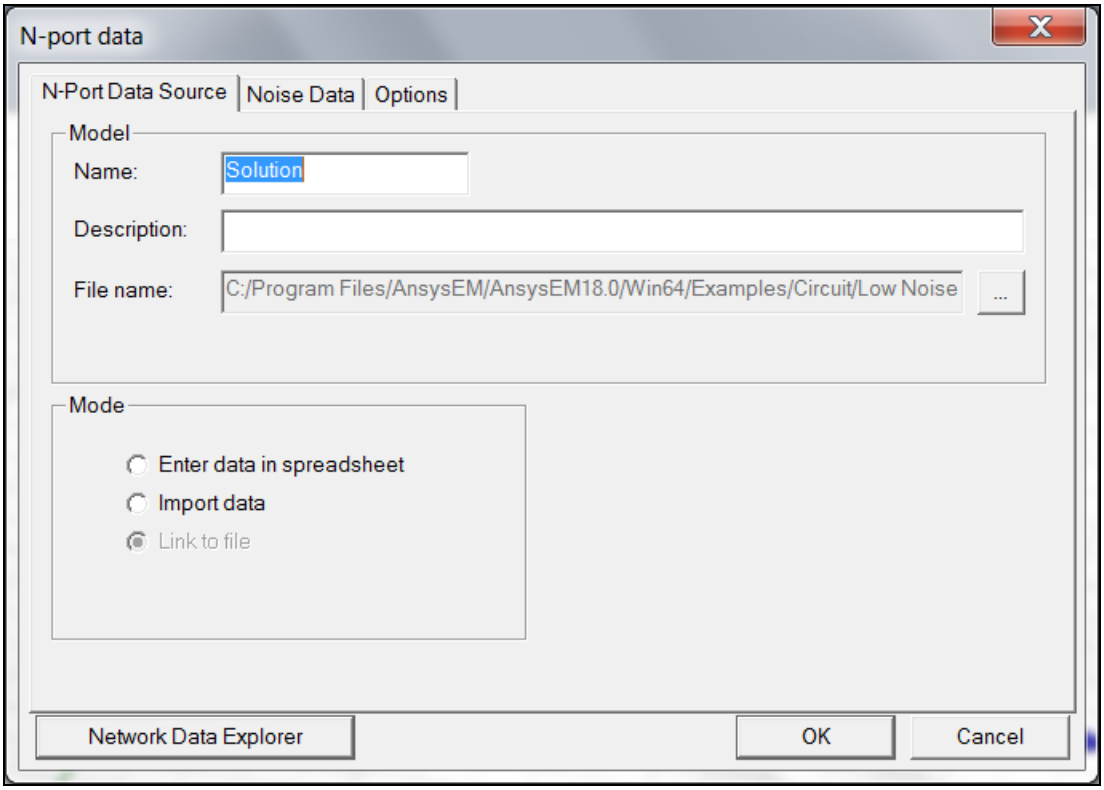
- Select **Circuit**> **Import Solution**
- From any design in the **Project Tree**, select **Analysis** > **Import Solution** to open the **Import Solution** window.



Importing Network Solution Data

If the imported solution includes network data (e.g., S-parameters), click **Network Data** on the **Import Solution** window. Then click **OK** to open the **N-port 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.



The **N-port data** window has the following tabs:

N-Port Data Source Tab

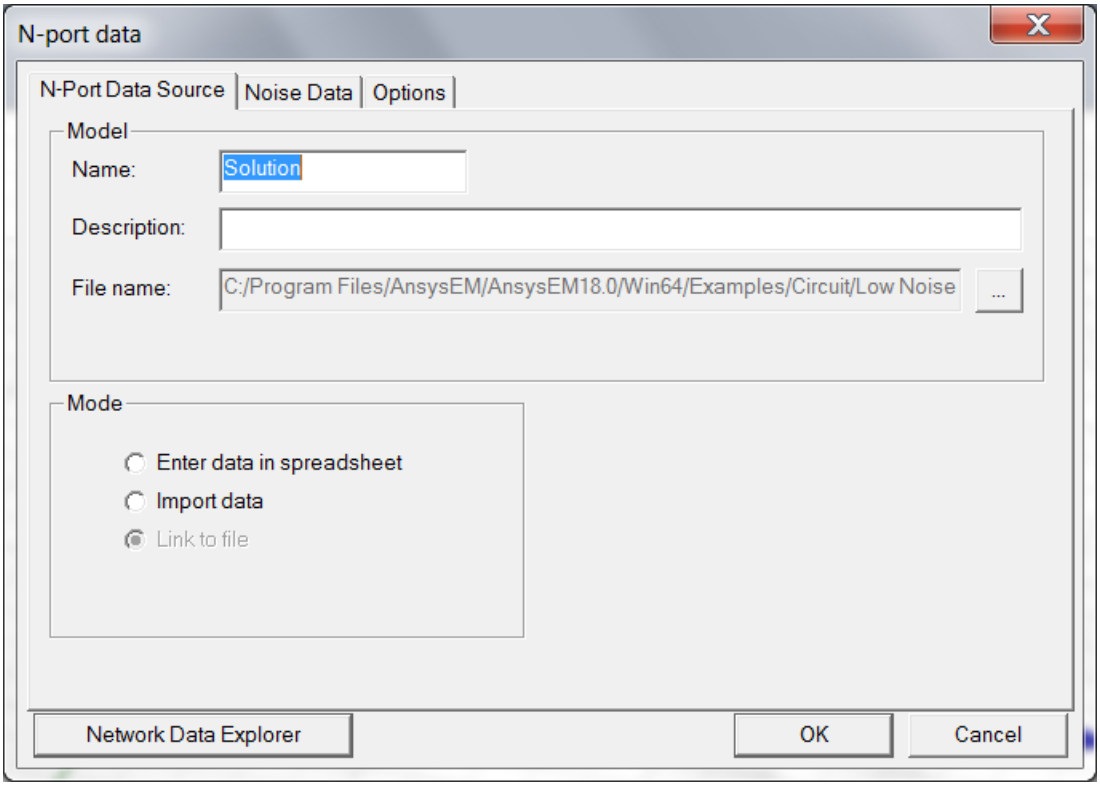
Noise Data Tab

Options Tab

N-Port Data Tab for Imported Solutions

The N-Port Data Source tab is displayed when the N-port data 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.



Name – Used to set or change the name of the imported solution as it appears in the **Project Tree**. To change the name, click in the box , then type the name you want.

Description — Used to set or change the description of the imported solution. To change the description, click in the box , then type the description you want.

File Name – Used to set or change the file name of the imported solution. To set the file name, click in the box and then type the file name you want, or click the **[...]** button to search for a file.

Multi-file selection — Used to import an S-element file.

Mode — The network solution data can be entered manually, copied into the project from an external file as **Import data**, or read from an external **Link to file** at analysis time. To manually enter solution data, click **Enter data in spreadsheet**, and enter the network data as described in the Noise Data Tab section.

To import an S-element file and copy it into the project from an external file:

1. Click **Import Data** in the **N-port data** window.
2. Specify the **File name** in one of the following ways:

- From the file name box, type the file name, including the full path, of the file that contains the network parameters you want to use.
- Click the **[...]** button to open the explorer window.

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

a. Select the appropriate file type from the **Files of type** area:
b. click the **Look in** field to browse to the directory containing the selected file.
c. Select the file in one of the following ways:

- Click the file's entry in the list.
- Type the file's name in the **File Name** box.

d. Click **Open**.

Once you have finished, the N-port data is copied into the project from an external file.

> **Note:**
>
> If you select multiple files using the Multi-file selection… and create a Text Array variable, the S-parameter data is NOT imported. It are the same as if you had selected the **Link to file** option.

To import solution data and copy it into the project from an external file:

1. Click **Import Data**.
2. Specify the file name in one of the following ways:

- From the file name box, type the file name, including the full path, of the file that contains the network solution.
- From the **File** group, select a file location option (Use Path, In PersonalLib folder, In userlib folder, or In syslib folder).
- Click the **[...]** button to open the explorer window.

a. From the explorer window, click the selected file type on the **Files of type** area.

— Touchstone (**.sNp**)

— HFSS v6.0 and higher (**.szg**)

— Touchstone (**.yNp**)

— Touchstone (**.zNp**)

— Touchstone (**.tou**)

— CITIfile (**.cit**)

— SSS file (**.sss**)

b. click the **Look in** field to browse to the directory containing the selected file.
c. Select the file in one of the following ways:

- Click the file's entry in the list.
- Type the file's name in the **File Name** box.

d. 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.

Once you have specified a file, view its contents in the Noise Data Tab.

> **Note:**
>
> S-parameter SSS datafile names must be of the format
>
> "<file name>_ <ports> .sss"
>
> where:
>
> <file name> is an alphanumeric string
>
> <ports> is an integer that specifies the number of ports
>
> such as:
>
> SparamFile_4.sss

To specify an external file from which network parameter data are read at analysis time:

1. Click **Link to file**.
2. From the **File** group, select a file location option (Use Path, In PersonalLib folder, In userlib folder, or In syslib folder).
3. Specify a file name in either of the following ways:

- From the file name box, type the file name , including the full path, of the file that contains the network parameters you want to use.
- Click **Browse** to open the **Open** window.
    a. Select the appropriate file type in the **Files of type** area.
    b. click the **Look in** field to browse to the directory containing the selected file.
    c. Select the file in one of the following ways:
        - Click the file's entry in the list.
        - Type the file's name in the **File Name** box.
    d. Click **Open**.

Once you have specified a file, view its contents in the Noise Data Tab.

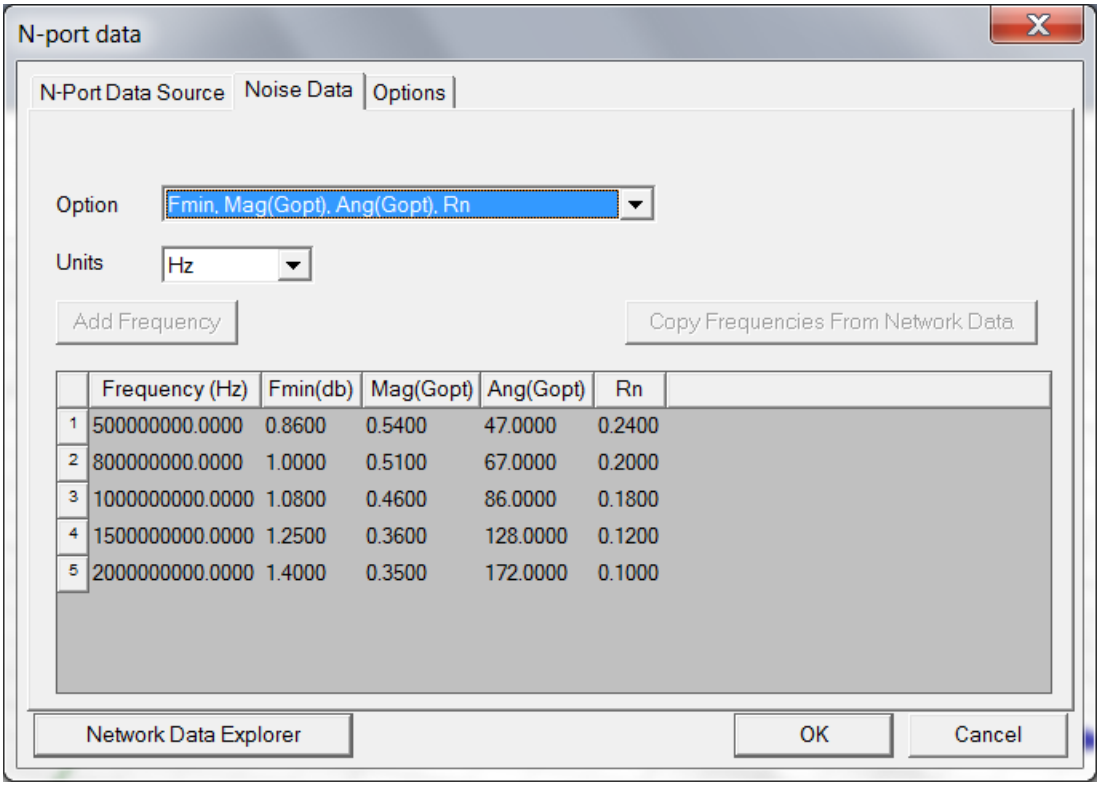
Noise Data Tab for Imported Solutions

The **Noise Data** tab can be used to enter solution data manually or used to view solution data from an imported file.

- To view the contents of an imported solution, double-click the solution in the **Project Tree** and select the **Noise Data** tab on the **N-port data** window.
- To manually enter solution data for post-processing, select **Enter data in spreadsheet**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 **N-Port Data** tab, then select the **Noise Data** tab on the **N-port data** window.



The following controls are available:

Domain – Sets the **Frequency** or **Time** domain.

Format – Sets the **Format** to one of the following: Magnitude/Phase, Real/Imaginary, or dB Phase.

Units – From the Frequency domain, sets the units of frequency: **Hz**, **KHz**, **MHz**, **GHz**, **THz**, or **rps**.From the Time domain, selects the units of time: **fs**, **ps**, **ns**, **us**, **ms**, or **s**.

S Matrix – Identifies the data as S-parameter values.

Y Matrix – Identifies the data as Y-parameter values.

Z Matrix – Identifies the data as Z-parameter values.

Gamma – Identifies the data as Gamma values.

Z0 – Identifies the data as Z0 values.

Ports – Sets the number of signal ports.

Phase Rotation — Click to set the **Phase Rotation** of the **N-Port Noise Data** source.

When the **Noise Data** options have been entered, click **OK** or select another tab.

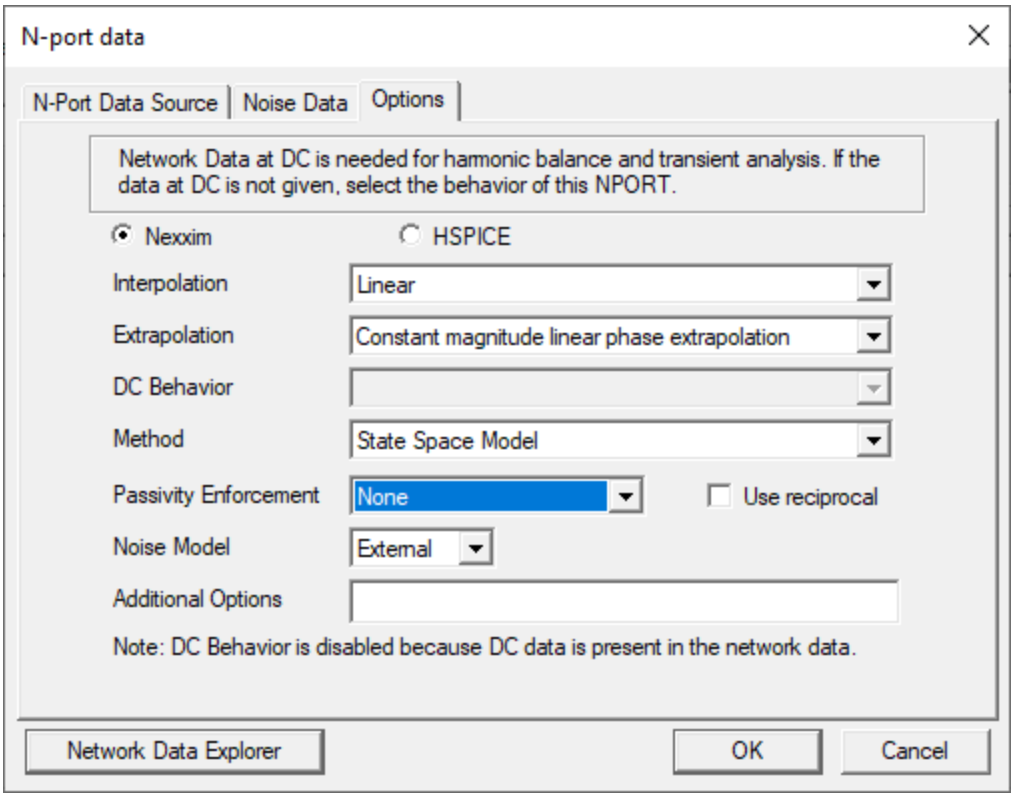
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

If the solution data file contains complex propagation constants or characteristic impedance values, select **Gamma** or $\mathbf{Z_o}$ to view the data — but to run a simulation in AEDT, the information is not required.

State Space N-Port Data Options Tab

The **Options** tab is used to select the behavior of the selected NPort.



The following controls are available:

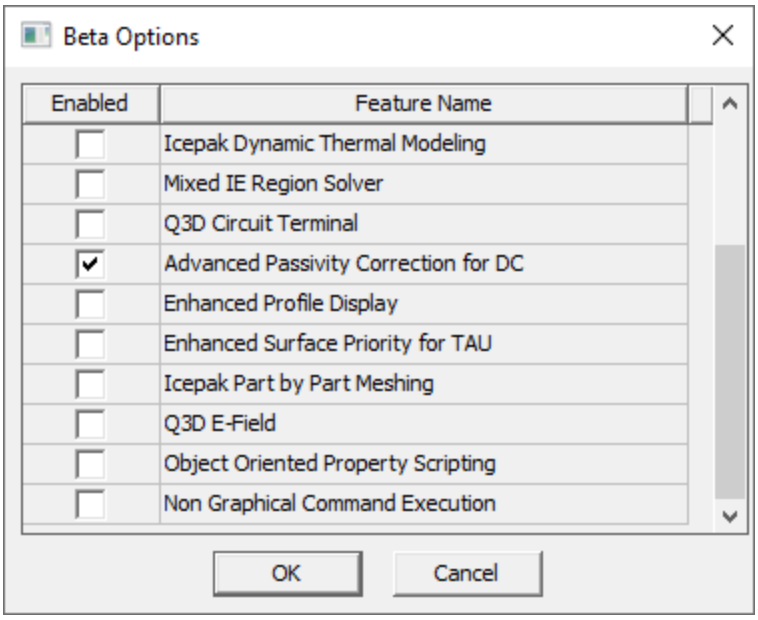
- **Nexxim/HSPICE** — Sets the NPort type. The selections for Interpolation, Extrapolation, DC Behavior, Method, and Passivity Enforcement depend on the NPort type selected.
- **Interpolation** — Specifies the interpolation method for data points in the frequency range of the Touchstone data.
- **Extrapolation** – Specifies the extrapolation method for data points outside the frequency range of the Touchstone data.
- **DC Behavior** — Specifies behavior of the N-Port at DC (zero frequency).

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

- **Method** – Specifies the method used to convert frequency domain data to the time domain. Choice of methods is available only for Nexxim Nports.
- **Passivity Enforcement** — Performs a passivity check on the S-parameter data and attempts to correct non-passive data. The options are None, IFPV, IFPVLF, Convex Optimization, and Passivity by Perturbation.

  Passive devices may dissipate or temporarily store energy, but never generate energy. Use the drop-down menu to select the method for enforcing passivity. Passivity enforcement is available only for Nexxim NPorts. See Passivity Checking and Enforcement. For details on passivity checking and enforcement, see S-Parameter Technical Notes.

  When applying IFPVLF, the passive DC fit to Scattering Parameter Data (S-data) can deteriorate to the point the inaccuracies of the (passive) fit to the corresponding Impedance Data (Z-data) at zero frequency (DC) and low-frequency may cause inaccurate time-domain steady state simulations. If preserving an accurate passive DC fit to S-data is important, like it is in power electronics applications, e.g., with DC-DC converters and power supply Voltage Regulator Module (VRM)), then click the **Advanced Passivity Correction for DC** beta option to enhance the passive DC fit to S-data using IFPVLF. The **Advanced Passivity Correction for DC** is located in the **Beta Options** in the Global Options.



- **Use reciprocal** – Computes the inverse, or reciprocal, at each frequency so the N-port can be used for deembedding. See Deembedding S-Matrices.

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

- **Noise Model** – Specifies noise model for DC and frequency-domain analyses:
  - **None** – No noise analysis
  - **Internal** – Use internal noise model
  - **External** – Use noise data from Touchstone file if present, else use internal noise model
- **Additional Options** — Allows you to add model-level options for Nexxim or HSPICE.

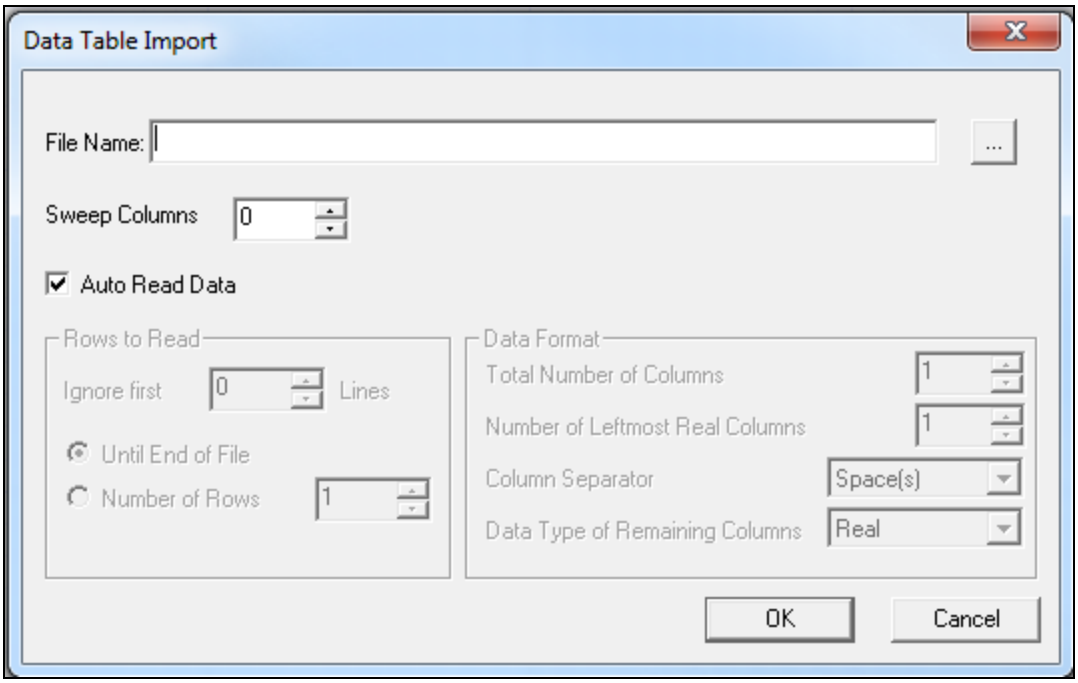
When the N-port Options have been entered, click **OK** or select another tab.

- **Network Data Explorer** – Allows you to view solution data by opening the Network Data Explorer.

When the N-port Options have been entered, click **OK** or select another tab.

Importing Tabular Solution Data

To import solution data in tabular format, select **Tabular Data** on the Import Solution window. (To specify the imported file should be read at analysis time rather than copied into the project, also click **Link to file**). Then click **OK** to open the **Data Table Import** window.



Enter the name of the tabular data file in the **File Name** window, or click the **[...]** button to look for the file. When you have entered all appropriate information into the **Data Table Import** window, click **OK** to import the data.

With Auto Read Data Selected

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

When **Auto Read Data** is selected, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform attempts to read the data in automatically, based on the following assumptions regarding the format of the tabular data:

- Rows that begin with a hash (#), star (*), exclamation point (!), or any non-number is ignored.
- Each comma, space, and tab is treated as a column separator.
- Numbers followed by an “i” or “I” are treated as complex values.
- The first column is always regarded as X data.
- All columns but the first are always regarded as Y data.
- Empty columns are zero filled.

With Auto Read Data Unselected

When **Auto Read Data** is not selected, the **Rows to Read** and **Data Format** subsections offer the following options:

Rows to Read

- **Ignore first** specifies the number of rows of data at the head of the file to ignore
- **Until End of File** specifies that all rows of data is to be read
- **Number of Rows** specifies the number of rows to be read

Data Format

- **Total Number of Columns** specifies the full number of columns of data
- **Number of Leftmost Real Columns** specifies the number of leftmost columns of real data
- **Column Separator** specifies the column separator: Space(s), Tab, Comma, or Period
- **Data Type of Remaining Columns** specifies the data type for the columns not included in the Number of Leftmost Real Columns:

Real	Real numbers
re/im	Complex numbers in real/imaginary format
mag/ang(deg)	Complex numbers in magnitude/angle format with angles in degrees
mag/ang(rad)	Complex numbers in magnitude/angle format with angles in radians

Sweep Columns

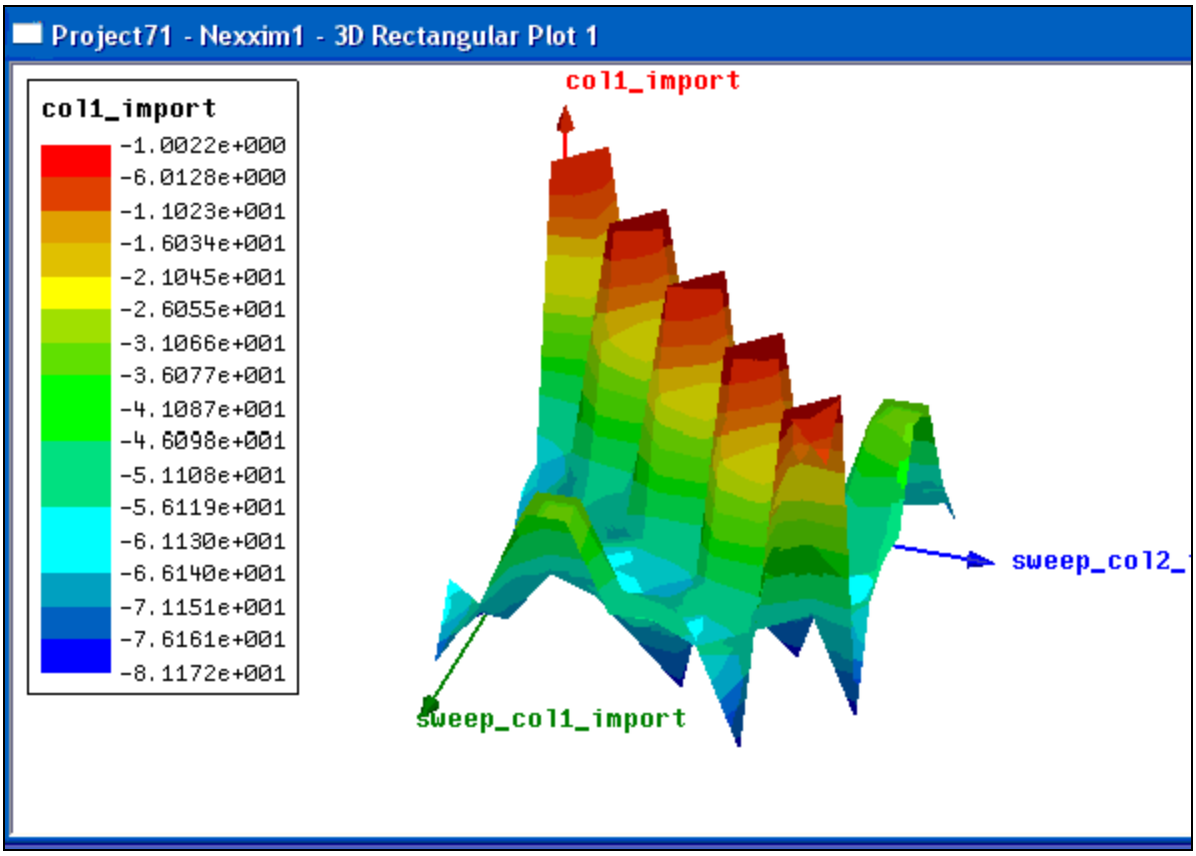
This option allows you to specify the number of leftmost columns in the imported file that can be treated as sweep data rather than quantity data. When **Sweep Columns** is non-zero, an implicit

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

sweep is created automatically as the file is read. For example, with **Sweep Columns** set to 2, the following imported data could be used to create the 3D Rectangular Plot shown in the following figure:

```
1 1 -11.9494901844939
1 2 -1.02976604277775
1 3 -65.0043850531982
1 4 -66.1547663779434
1 5 -76.5001257180356
1 6 -77.0620478855849
1 7 -78.7052860174241
1 8 -71.4292118632088
1 9 -59.271550467952
1 10 -73.4397348975777
2 1 -1.02976604277775
2 2 -13.7362474109939
2 3 -64.4351612784164
2 4 -63.9486697204035
2 5 -75.69071743575
2 6 -74.7859772914394
```


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

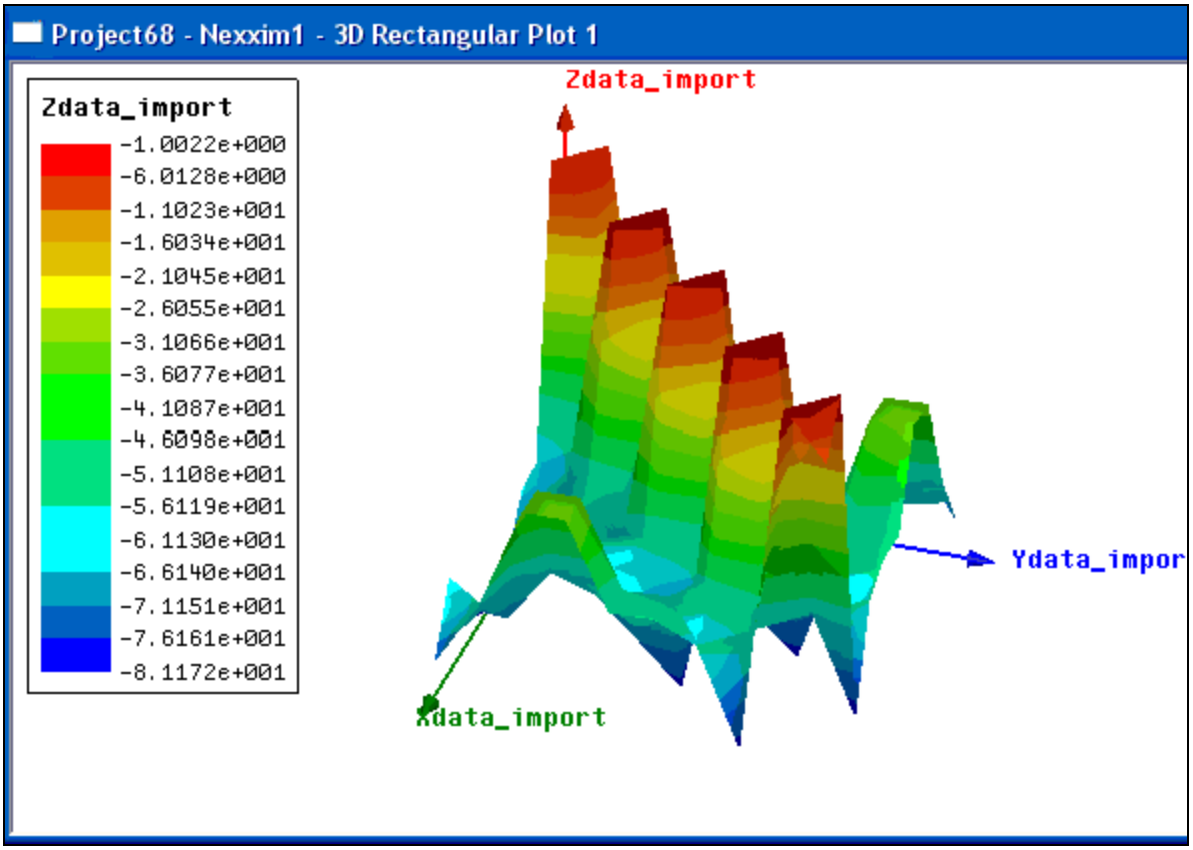


From the imported file, the line immediately preceding the start of numerical data is checked for column names. If this line contains the selected number of entries, these entries are then used as column-data annotations in the generated plot, as shown in the following example.

```
Xdata Ydata Zdata
1 1 -11.9494901844939
1 2 -1.02976604277775
1 3 -65.0043850531982
1 4 -66.1547663779434
1 5 -76.5001257180356
1 6 -77.0620478855849
1 7 -78.7052860174241
1 8 -71.4292118632088
1 9 -59.271550467952
1 10 -73.4397348975777
2 1 -1.02976604277775
2 2 -13.7362474109939
2 3 -64.4351612784164
```


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

Each column name (Xdata, Ydata, Zdata) is appended with the string “import” before being added to the plot.



Post-processing Imported Solution Data

To generate a report on the imported solution data, do the following:

1. Expand **Analysis** in the Project tree. Verify the presence of the imported solution.
2. Click on **Results** in the Project tree and select **Create Report**. Select the report type and display type on the **Create Report** window,, then click **OK**.

From the **Traces** window, select the imported solution on the **Solutions** drop-down menu. Complete the data in the **Traces** window, then click **Done**. The **Report** window opens to display the imported solution.

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

6 - Geometry in HFSS 3D Layout

HFSS 3D Layout supports a variety of multi-layer structures and components that can be placed in those structures. Among the geometries are:

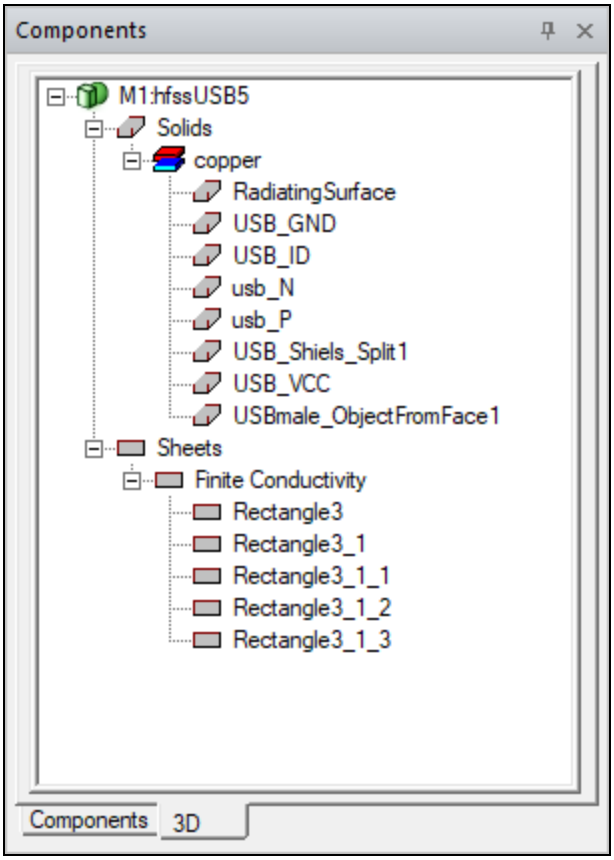
- Stride Lightweight Geometry
- Component Placement in HFSS 3D Layout
- Hierarchical Design
- DCIR and Icepak Simulation on Hierarchical Designs
- 3D Component Library
- Stackup Definitions
- Programmatic EDB Cells
- Setting Uniform Temperature

Stride Lightweight Geometry in HFSS 3D Layout

Stride is a lightweight 3D geometry representation used for 3D CAD components within HFSS 3D Layout. Formats such as IGES and STEP can be imported, at which time they are converted to Stride for lightweight management.

After importing components, view them on the **3D** tab of the **Components** window.

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



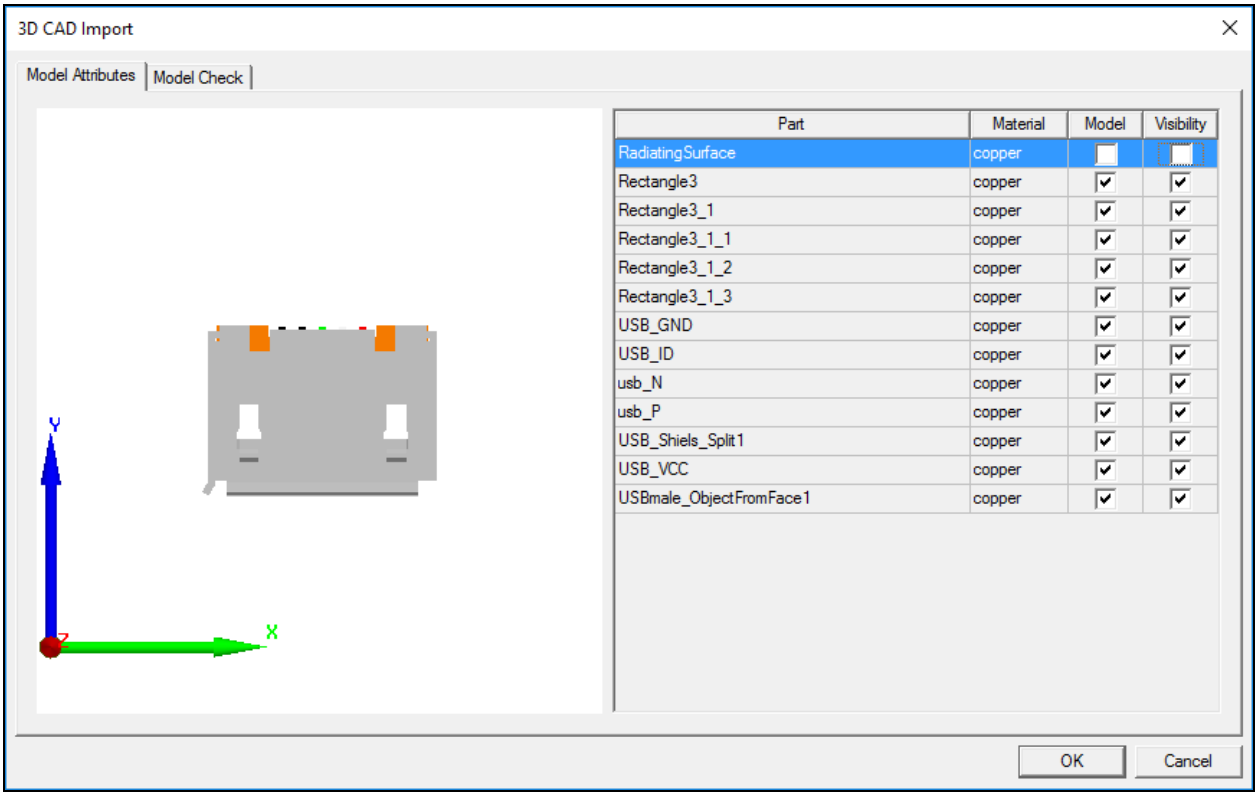
Selecting a component or part within a component highlights it in the design area. You can then edit certain properties in the **Properties** window and trace physical connections between components.

Importing 3D CAD Components into HFSS 3D Layout

To import a 3D CAD component into HFSS 3D Layout:

1. Click **Layout** > **Import** > **3D CAD**.
2. From the explorer window, click the component's file and click **Open**. The **3D CAD Import** window opens, on the **Model Attributes** tab.

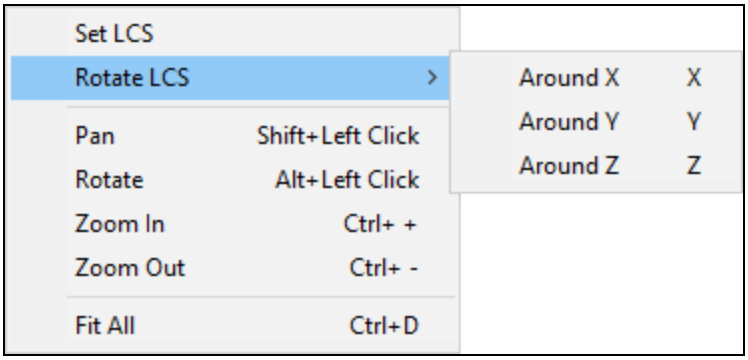
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left pane contains a graphic representation of the MCAD component.

The right pane lists the component's parts and their materials.

3. Right-click in the left pane to open a shortcut menu.



From here, you can perform the following actions:

- **Set LCS** – turns the cursor into a crosshair that allows you to click a face, edge, or vertex and reposition the local coordinate system to that 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 orientation of the Z axis is based on face normal. Placement of an MCAD component is simplified if a logical local coordinate system for the MCAD component is selected (e.g., select the center of a face that must be coincident with a pin in HFSS 3D Layout; when placing the MCAD model, the LCS origin snap to geometry based on the current snap settings).

To exit placement mode, right-click and deselect **Set LCS**.

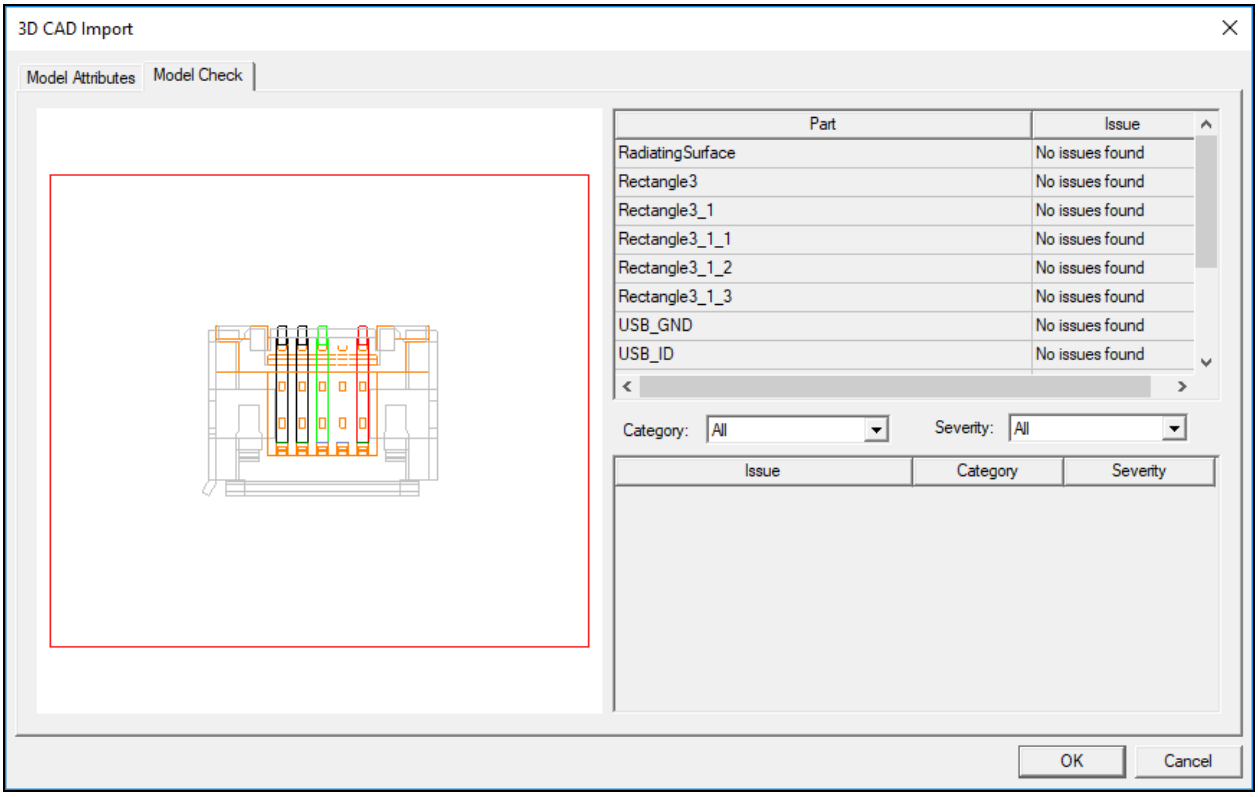
- **Rotate LCS** – reorients the local coordinate system by rotating it about the selected axis. Also press the **X**, **Y**, or **Z** keys.
- **Pan** – allows you to move the component in the pane. Also use **Shift**+**click**.
- **Rotate** – allows you to rotate the component in the pane. Also use **Alt**+**click**.
- **Zoom In / Zoom Out** – allows you to zoom the component in the pane. Also use **Ctrl**+**E** and **Ctrl**+**F**.
- **Fit All** – fits the component to the pane. Also use **Ctrl**+**D**.

> **Note:**
>
> Viewing operations only affect the display in the **3D CAD Import** window. They do not change the local coordinate system.

4. Click the material field to enable an **Edit** option, which allows you to change the material or its properties.
5. Click the **Model** check boxes to select and deselect parts for modeling, and the **Visibility** check boxes to alter the visibility of parts.
6. Click the **Model Check** tab.

   The **Model Check** 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.



The **Model Check** tab performs sanity checks on the MCAD component and displays the results. If any issues are found, you can sort them using the **Category** and **Severity** drop-down menus.

7. Click **OK** to import the component.

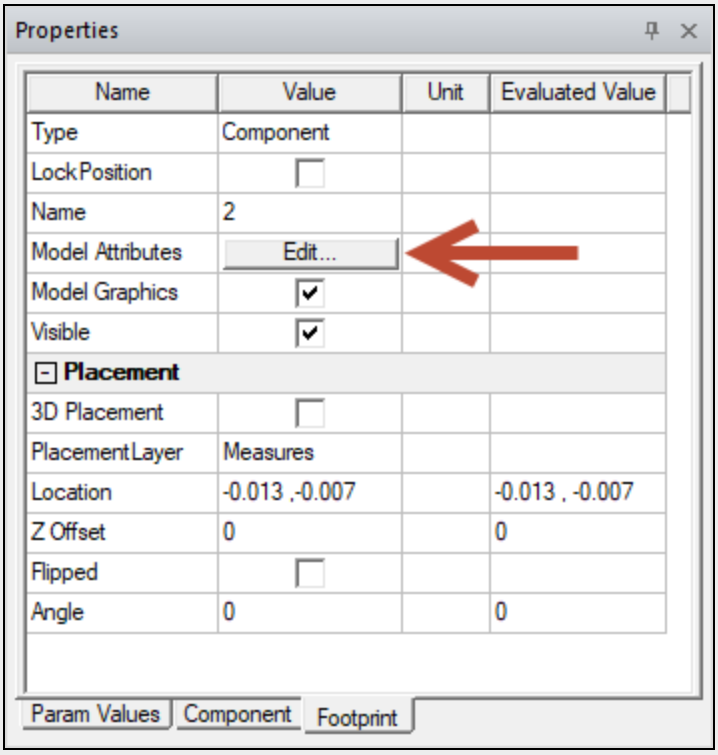
   The imported component appears in the **3D** tab of the **Components** window.

8. From the **Layout Editor**, click where the component should go.

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

Important:

If you need to change the component properties after import, you can do so by selecting the component and clicking **Edit** on the **Footprint** tab of the **Properties** window:

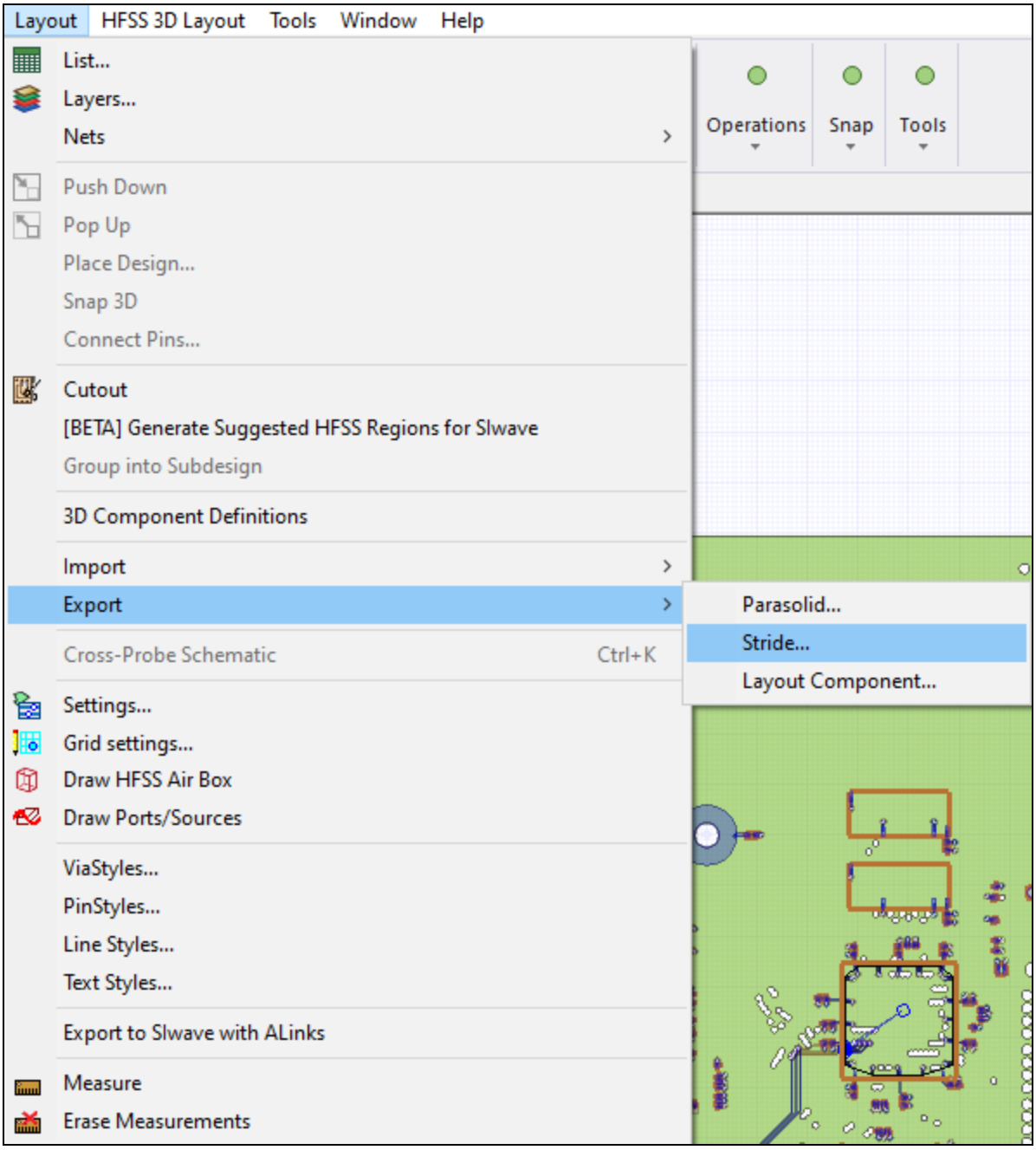


Exporting HFSS 3D Layout to Stride Format

HFSS 3D Layout can export Stride projects (*.project) files. Follow these steps to export a **Stride .project** file.

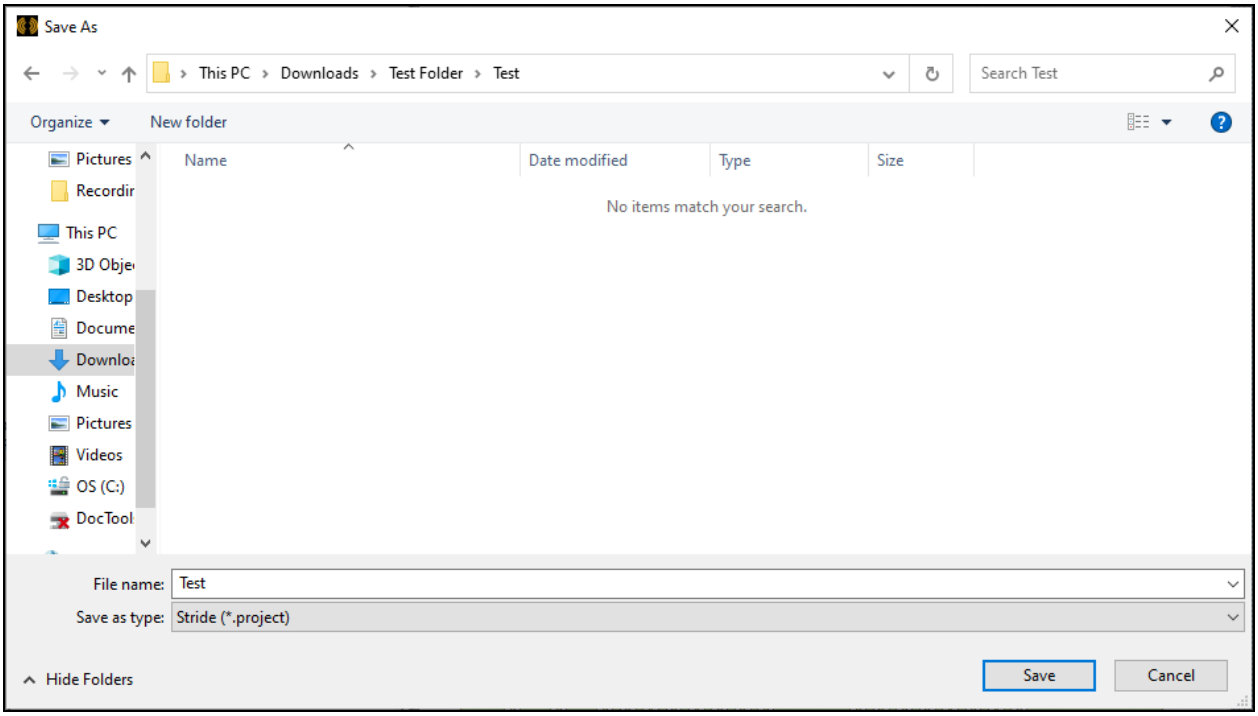
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From **Layout**, select **Export** > **Stride** to open an explorer 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. Navigate to a selected directory to save the project.



3. Enter a selected **File name**.
4. Click **Save**.

Adding HFSS 3D Components to HFSS 3D Layout

HFSS 3D Layout allows for the use of HFSS 3D Components with the HFSS solver. Placement, setup, and options are similar to working with an HFSS 3D Dynamic Link in **Layout**.

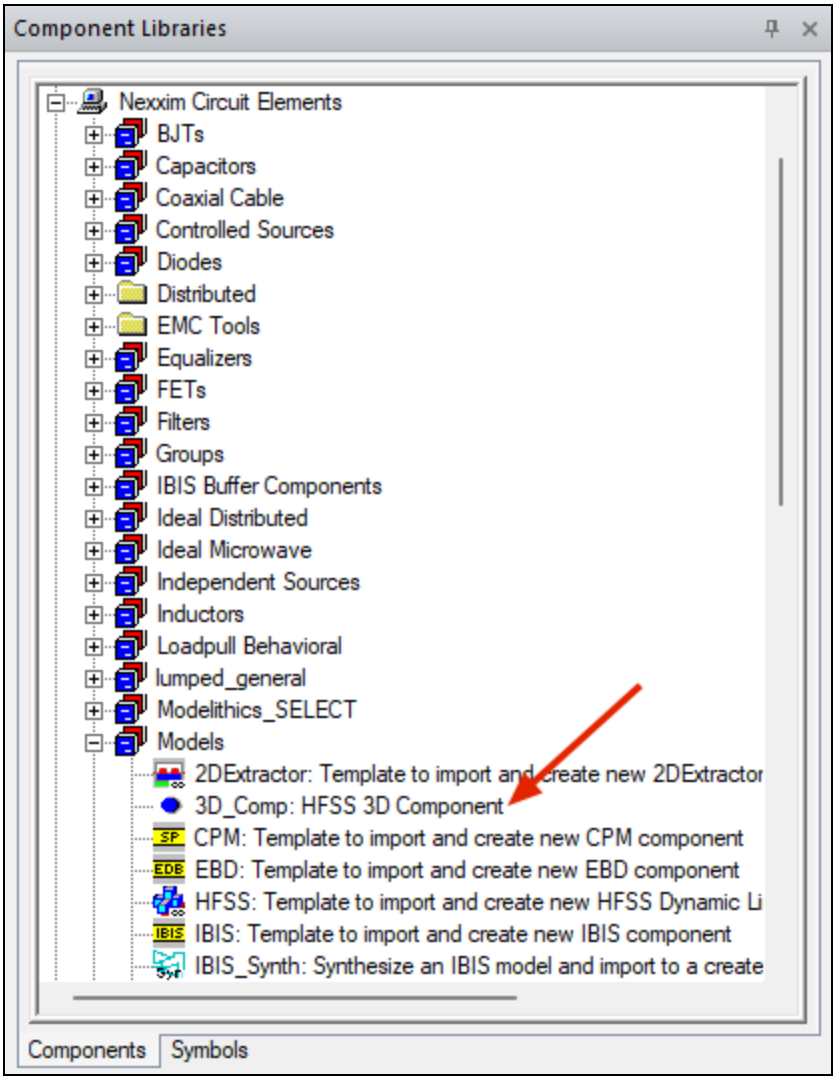
To import a 3D component into HFSS 3D Layout, either **click+drag** a *.a3dcomp file into an HFSS 3D Layout design or perform these steps:

1. Click **Layout** > **Import** > **3D Component**.
2. From the **Select Model File** window, navigate to and select the component's file and click **Open**.
3. From the **Layout Editor**, click where the component should be located. The imported component appears in the **3D** tab of the **Components** window.

To add a 3D component on the Component Libraries:

1. Open the **Component Libraries** window.
2. Double-click **Nexxim Circuit Elements** > **Models** > **3D_Comp: HFSS 3D Components**.

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 **Select Model File** window, click the component's file and click **Open**.
4. From the **Layout Editor**, click where the component should be located. The imported component appears in the **3D** tab of the **Components** window.

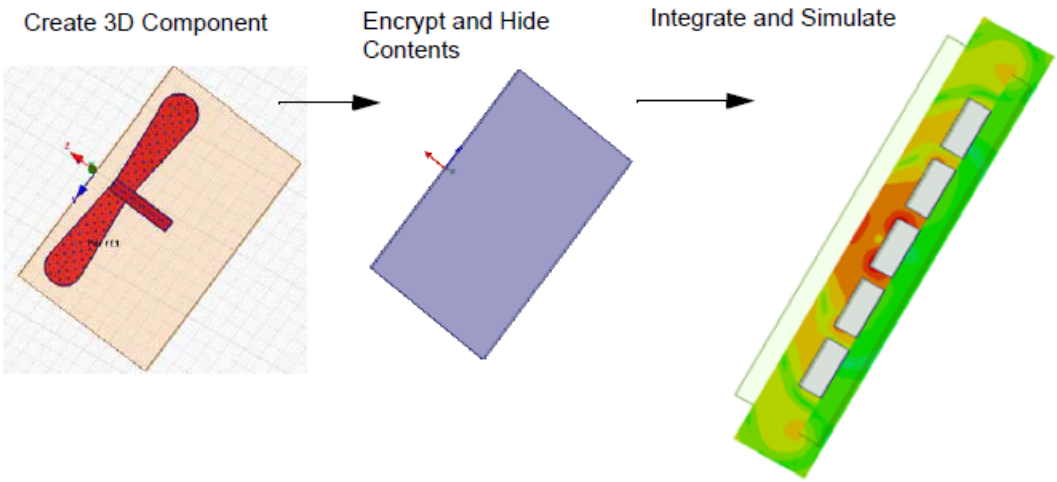
If a parameter was added to the 3D component in HFSS, the parameter appears on the **Param Values** tab of the **Properties** window where it can be updated.

3D Component Library

The 3D Modeler lets you access predefined 3D component libraries and custom 3D components using:

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

- 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.

In HFSS 3D Layout, custom 3D components must be created in HFSS and then imported into HFSS 3D Layout. 3D components appear in the **3D** tab of the **Components** window. Once in HFSS 3D Layout, the 3D components cannot be edited except for changes to their parameters. HFSS 3D Layout only supports 3D components with Driven Terminal solution types. In HFSS, this solution type is **Terminal** in the **Solution Type** dialog box.

A Beta Feature for **Parallel Component Mesh Adapt** improves the load balance across components by assigning different core counts and memory allocations for different size components. The workflow is applicable to all parallel component adapt features. Without the beta feature, all the components are assigned the same number of cores and same amount of memory during component adapt.

Local variables can be assigned to a 3D component's parameters in HFSS 3D Layout and then used in parametric simulations. Change the parameters in the **Param Values** tab of the **Properties** window.

HFSS 3D Layout supports encrypted 3D components and 3D components with hidden content. If a 3D component has hidden content, it appears as **Encryption Settings** on the **3D** tab. Use the **3D** tab to select a 3D component when all contents of its component are hidden. If possible, keep something such as the bounding box visible in the 3D component. If nothing is visible, 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 still select the bounding box, however you won't be able to see any part of the component. The parameters and excitations of a 3D component are visible even when all contents of the component are hidden.

The documentation includes the following sections:

Using the Component Libraries Window

Properties of 3D Components

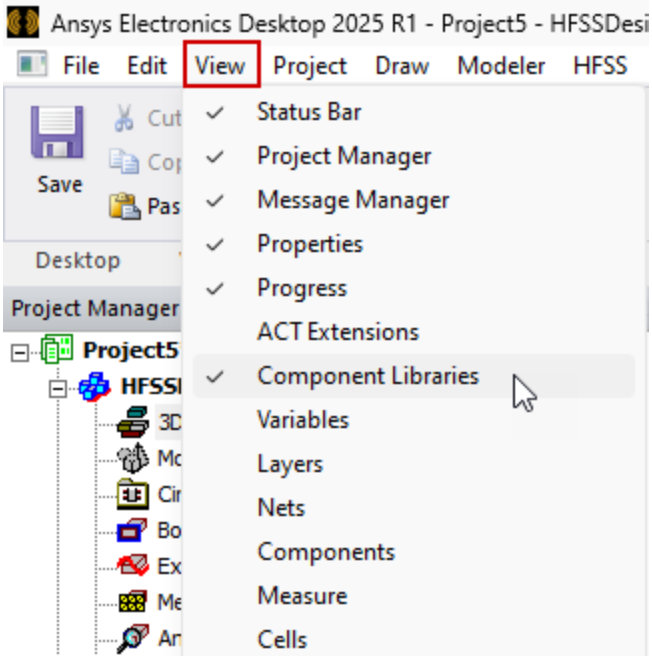
Editing 3D Component Properties

3D Component Agents

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 .

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.



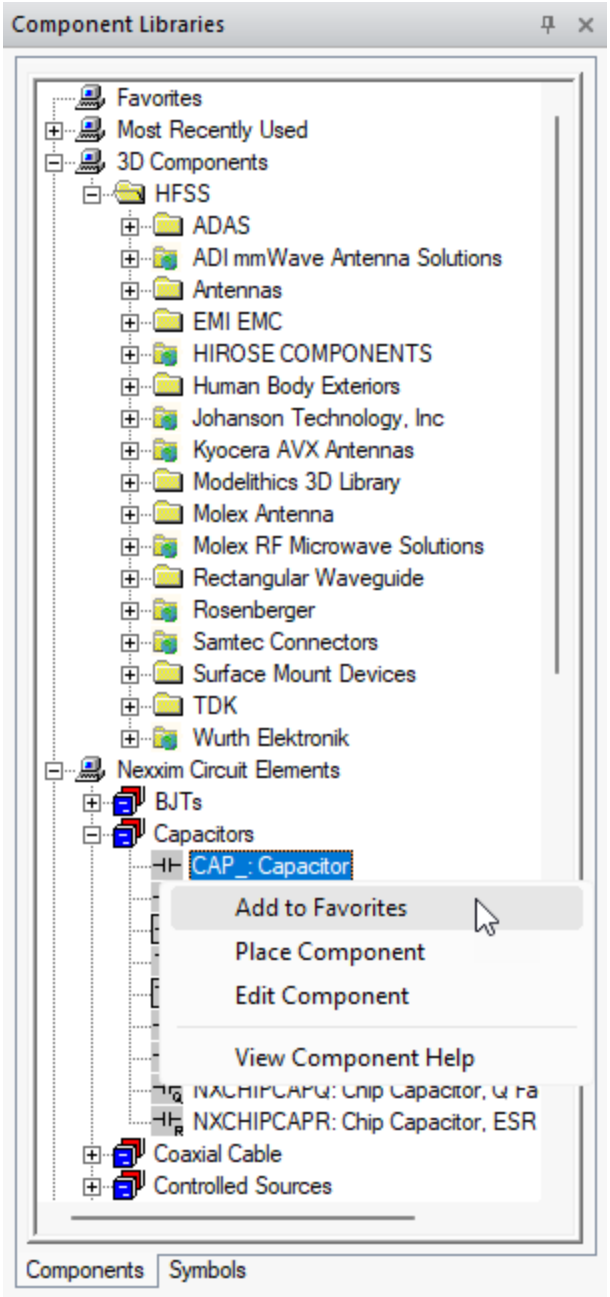
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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. For HFSS 3D Layout designs, select this folder from the **Filter** drop-down menu.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. The component is inserted with default component parameter values and is placed in the currently active coordinate system.

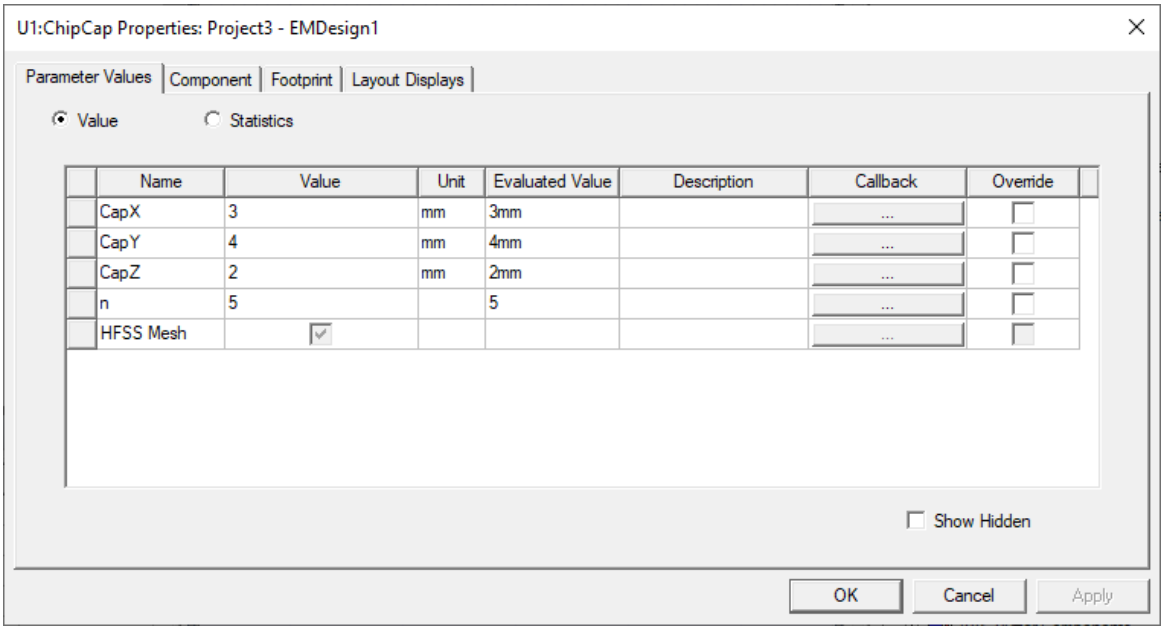
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 HFSS 3D Layout should be organized under 3DComponents\HFSS 3D Layout, 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, HFSS 3D Layout will ignore component files placed directly under the Maxwell subfolder.

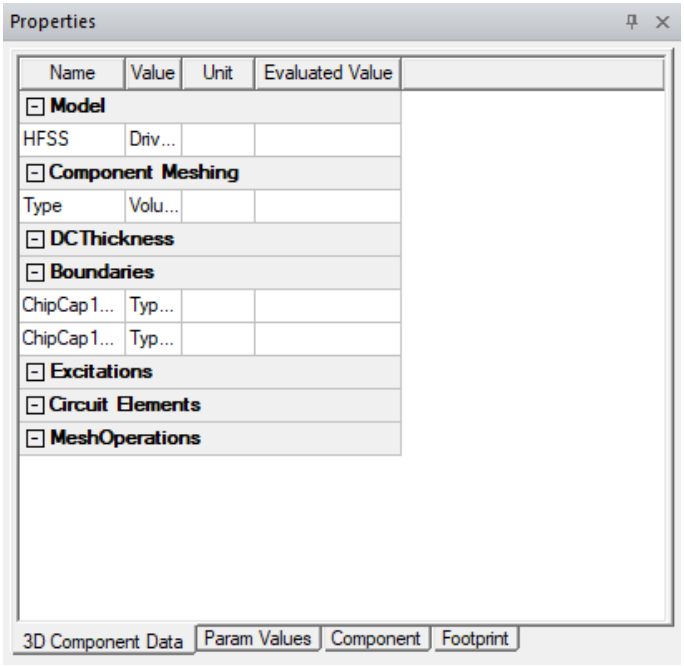
Viewing Properties of 3D Components in 3D Layout

The Properties displayed for a selected 3D component depend on how you select that component. For a complete display of Properties, double click the selected component in the **Layout** editor. Alternatively, click **Edit** > **Properties** using the menu bar. This action displays a window with tabs for all component properties.



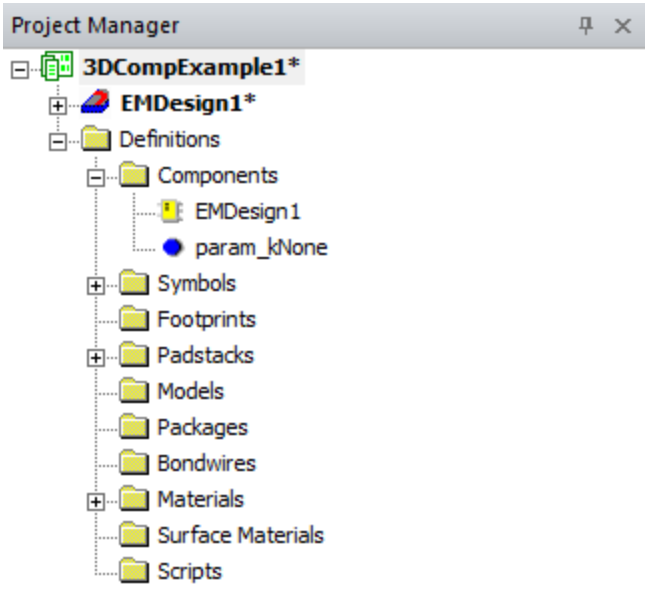
Selecting a 3D component in the **Layout** editor displays tabs for **3D Component Data**, **Param Values**, **Component**, and **Footprint** in the docked **Properties** window.

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



3D Components in the Project Manager Window of HFSS 3D Layout

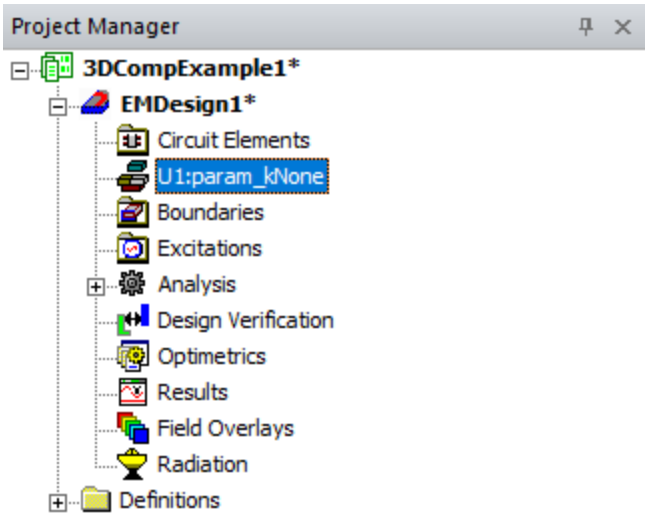
The **Project Manager** window organizes any 3D components added to the HFSS 3D Layout design under the **Components** folder in the **Project Tree**.



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

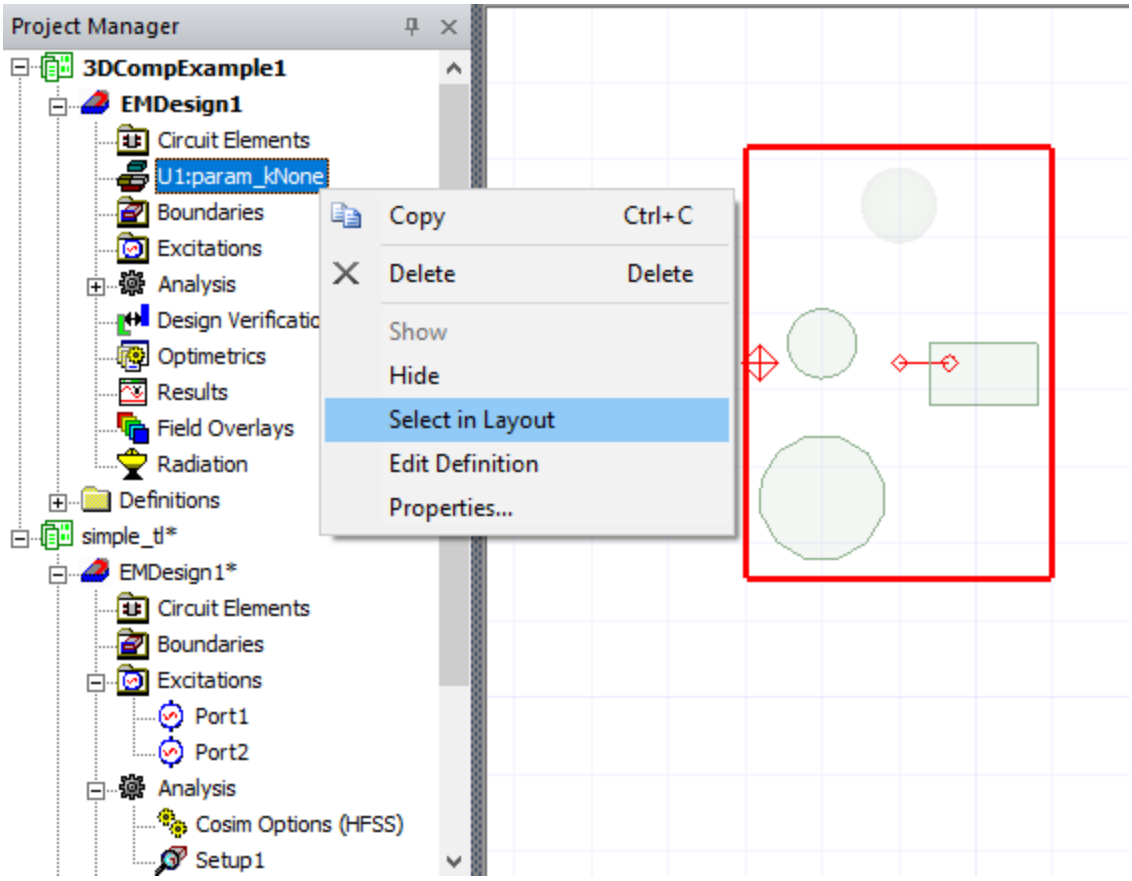
Click + drag a component to the **Layout Editor** to add it to the active design. Double-click a component in the **Project Tree** to open the Edit Component window.

Once a component is placed in the **Layout Editor**, it also receives a subfolder in the active design folder.



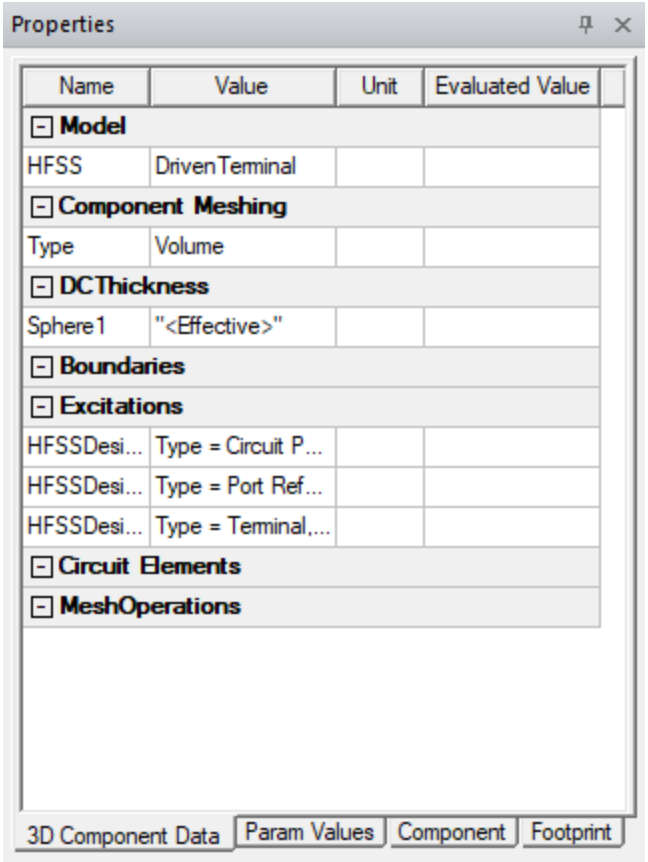
Right-click the 3D component in the active design folder and select **Select in Layout** to outline the placed component in red.

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



Boundaries defined inside the component do not appear in the **Project Manager**. However, boundary properties and other instance attributes will appear in the **Properties** window. Select the 3D component in the **Layout Editor** or right-click the component in the **Project Manager** > **Project Tree** > active design folder and the **Properties** window will populate with all of the component's instance attributes (e.g., find the component's defined boundaries under the *3D Component Data* 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.

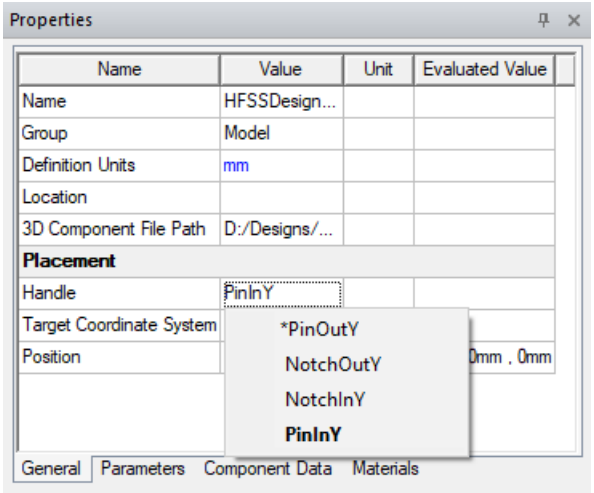


Editing 3D Component Properties

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.

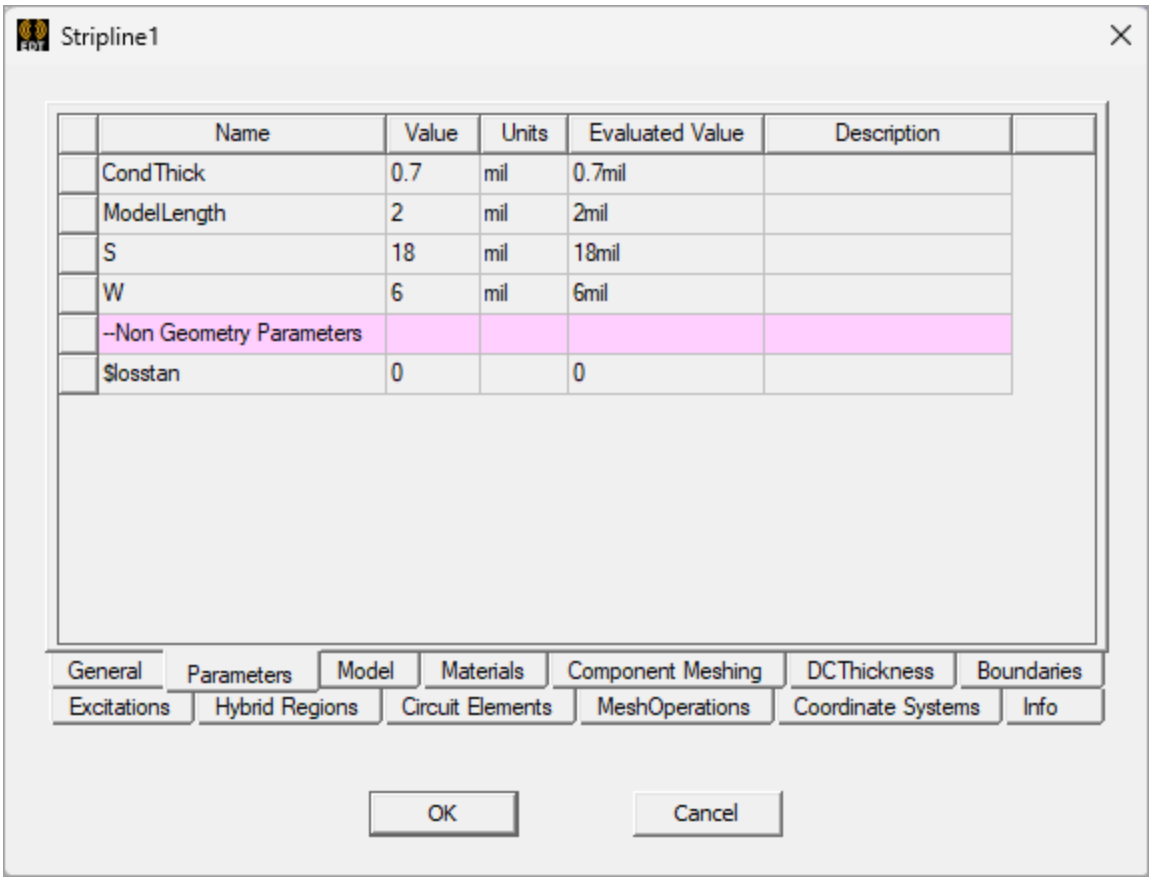
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **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.

- 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.



- Copy/Paste component
- Delete component
- Arrange (Move, Rotate, Mirror) component
- Duplicate component (Along Line, Around Axis, or Mirror)
- Edit Attributes for Display Wire frame, Color, and Transparency.
- Use **Edit** > **Select** > **Submodel** to quickly select component
- Copy/Paste component
- Delete component
- Arrange component: Move, Rotate, and Mirror
- Duplicate component: Along Line, Around Axis, and Mirror
- Edit Attributes for Display Wire frame, Color, and Transparency
- Use **Edit** > **Select** > **Submodel** to quickly select component

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

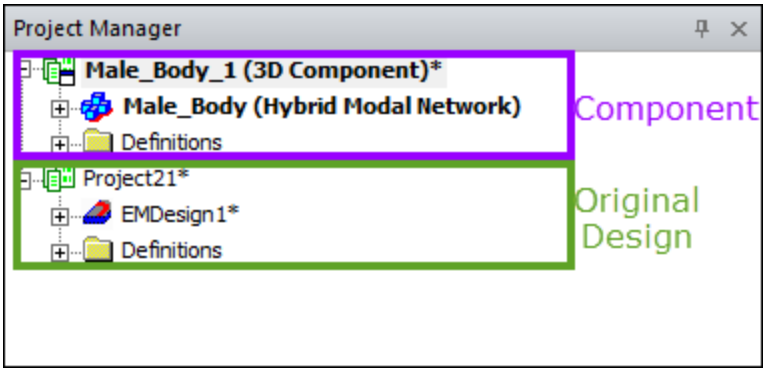
Editing a 3D Component Definition

> **Note:**
>
> If you reached this page looking for help on editing the definition of a **Layout Component**, as opposed to a *3D Component*, go here instead.

The **Edit Definition** feature allows the user to create new component definition using an existing, unencrypted 3D component file. This speeds up the task of creating new versions of components without having to start from the beginning. When **Edit Definition** is initiated, **Electronics Desktop** automatically creates a new project containing all 3D component data from the file in 'exploded' form. Component materials are added at the project level; variables are added at the project and design level; a detailed history is created for all objects in the component; and design data is added to the appropriate folders in the Project Manager.

> **Important:**
>
> Only 3D components that are **not encrypted** can be edited as described in this section.

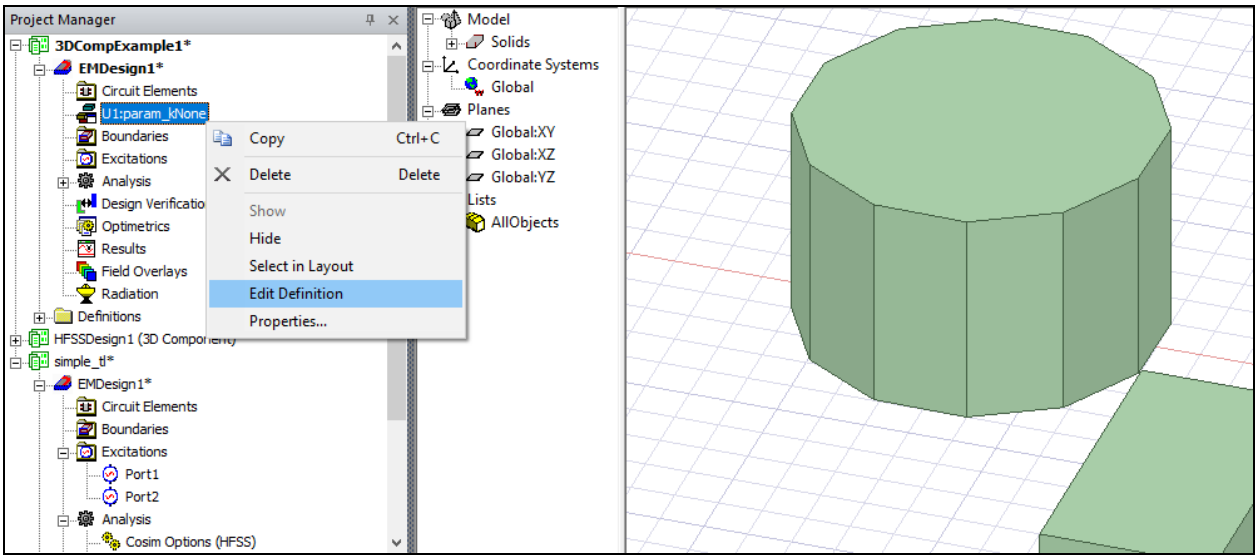
The **Edit Definition** command creates a new project, named for the component, in the **Project Manager** window.



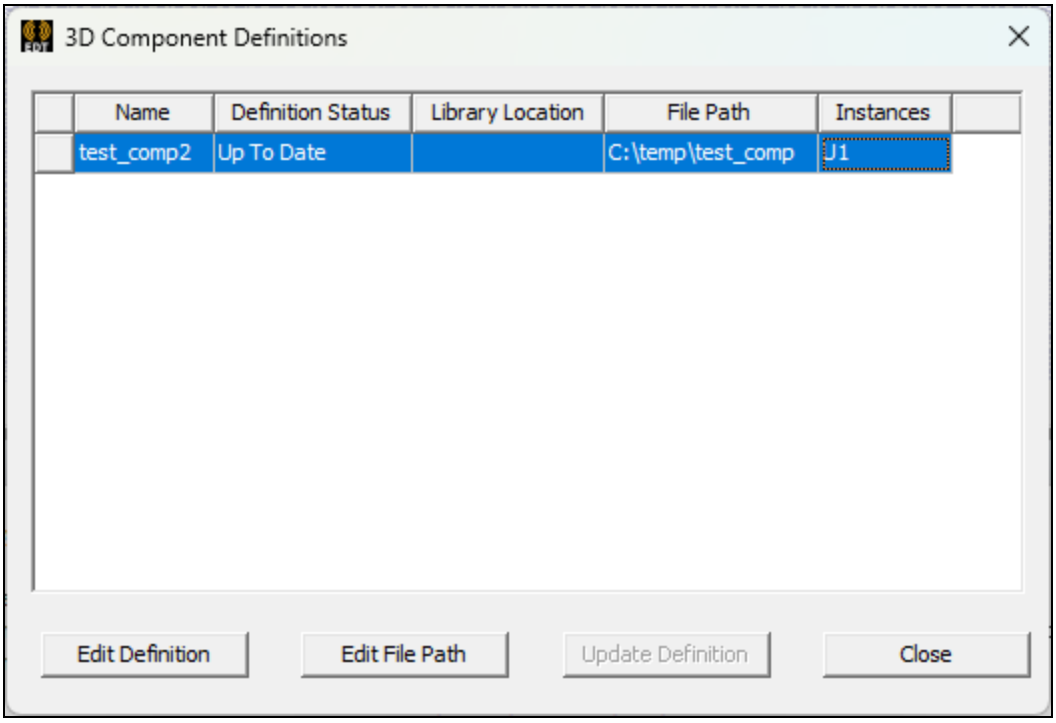
When the edited component is saved, an **Update 3D Component** window appears. From the **Update 3D Component** window, choose whether to save the edited component as a local instance or to the original library. To edit a 3D component definition, do one of the following:

1. Navigate to the **Edit Definition** window by following one of the following paths:
   - From the **Project Manager**, expand the **Project Tree** and active design folder. Then right-click the 3D 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.



- From the **Layout Editor**, right-click the 3D component and select **3D Components** > **Edit Definition**.
- Click **Layout** > **3D Component Definitions** to open the **3D Component Definitions** window. From the **3D Component Definitions** window, select a component and click **Edit Definition**.

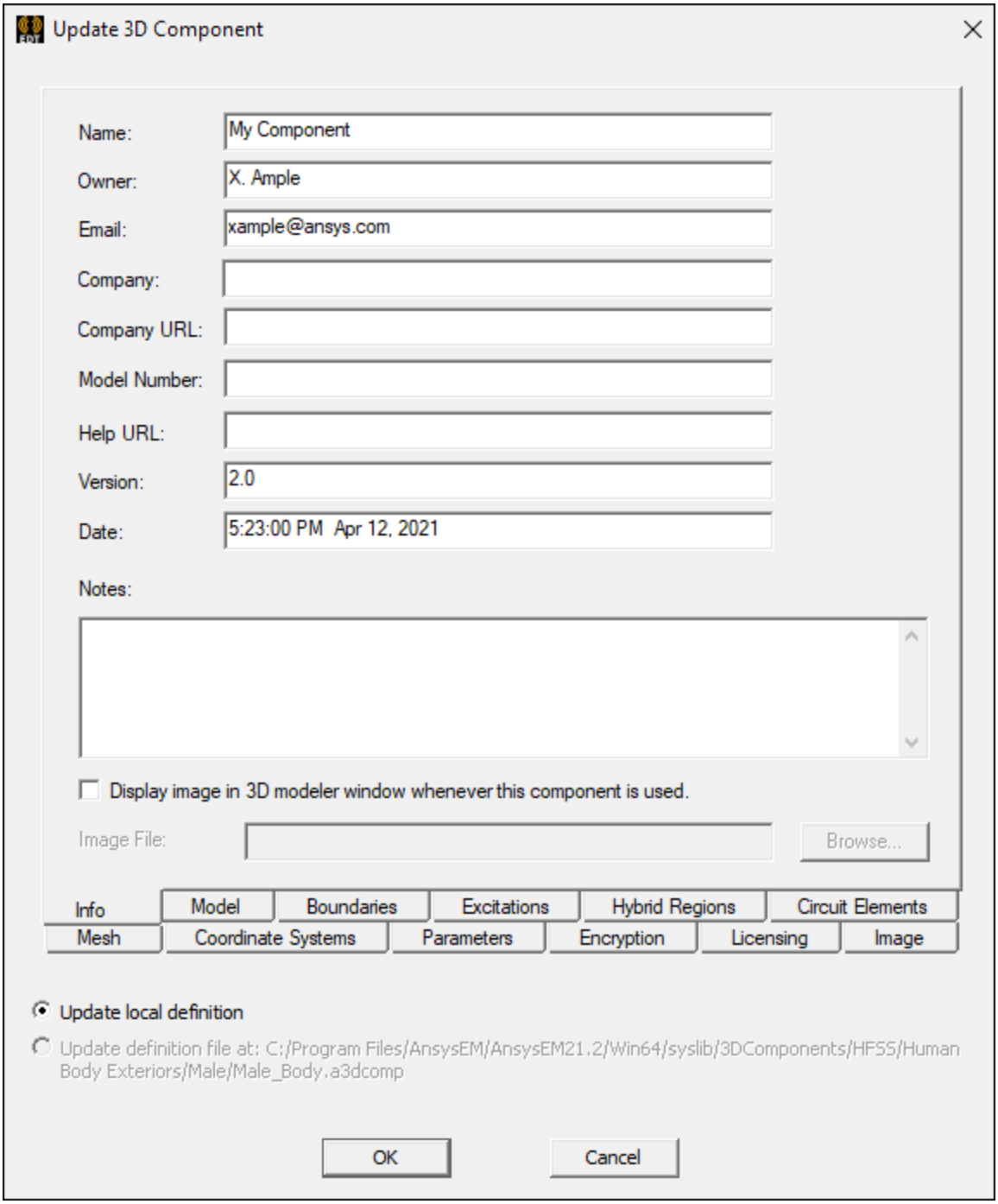


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

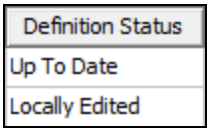
6. Click **OK**.

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**.
   - Click **Layout** > **3D Component 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**.
   - Click **Layout** > **3D Component Definitions**.

   Components with updates available display "Update Available" in the **Definition Status** column.

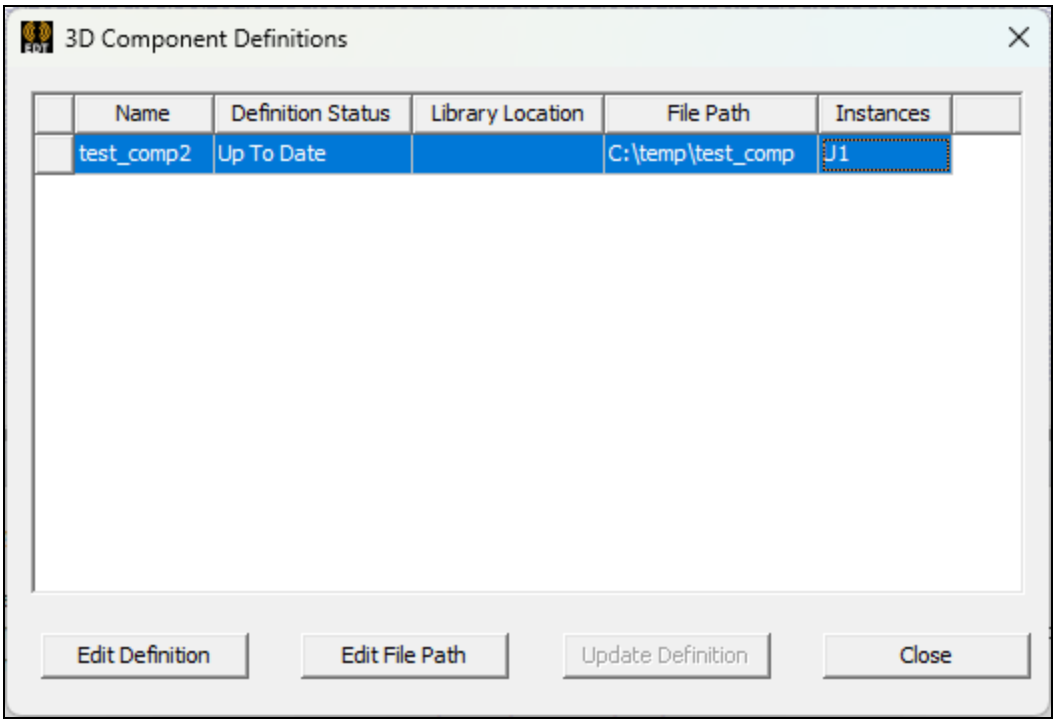
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the desired component and click **Update Definition**.
3. Click **Close** to exit the window.

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.

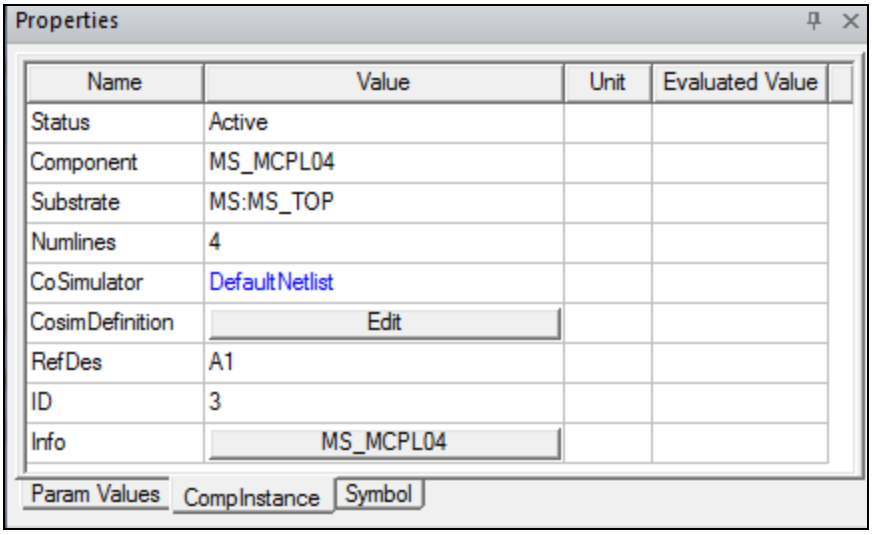
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stackup Definitions for HFSS 3D Models

The **Stackup** field on the **Define Co-simulation** window for an EM model is set to one of the following three values, in order to indicate which stackup definition are used to model a component when it is simulated by one of the EM solvers: Substrate Stackup, Layout Stackup, Footprint Stackup.

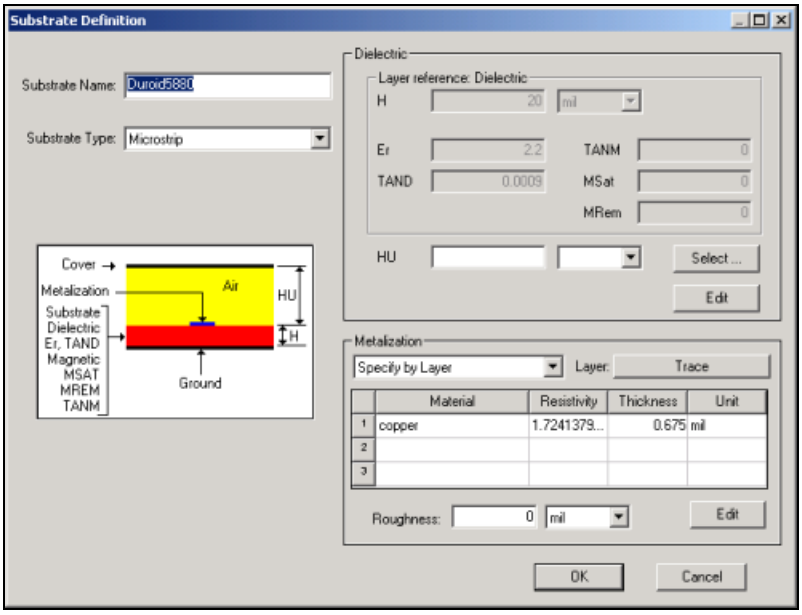
Substrate Stackup

By default, when a component is placed, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform automatically associates a substrate definition with the CompInstance. Not all components need substrate definitions. If it is present, the **Substrate** definition are shown in the Substrate property in the **CompInstance** tab.



The **Substrate** property menu shows all defined substrates for this Circuit design. The menu may include the choice **<none>** to allow changing to a component that doesn't use a substrate definition (See Using Component Groups). The menu contains the **New** command which opens the **Substrate Definition** window.

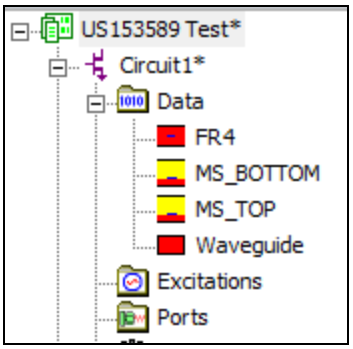
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this window, you have the option of either manually entering data describing the substrate or you can specify the data be taken from layers defined for the layout stackup (See Layout Stackup).

- To manually edit the Dielectric, click **Edit**.
- To pull data from a dielectric layer, click **Select**.
- To manually edit the metalization, click **Edit**.
- To pull data from a metalized layer, change the **Trace Metalization** drop-down menu to **Specify by Layer**, then click **Layer** to select the layer.

Substrate Definitions are all listed in the **Project Manager** window under the **Data** folder for the Circuit design.



Double-click one of these icons to open the **Substrate Definition** window and edit the definition.

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

HFSS Etch Factor

Substrate manufacturing does not create exact geometry. The actual geometry of substrate manufacturing often results in different thicknesses in the dimensions of each geometry within a layer — and that difference is represented by an “etch” or “etch factor”. HFSS geometry allows for the presence of an etch factor in the design and modeling of different substrate layers. The specific etch factor that applies to a particular layer is usually available on the manufacturer. But in general, the etch factor is calculated by taking the ratio of the etch depth to the undercut, where the undercut is defined as being the part of the wafer that is etched away under the photoresist:

```
etch_factor = layer_thickness / (bottom_dimension - top_dimension) /
2
```

- When the bottom dimension of the stackup is larger than the top, the etch factor is positive.
- When the top dimension is larger than the bottom, the etch factor is negative.

For example, with a positive etch factor, a transmission line with a 2mm nominal width are 2mm wide on the bottom and narrower at the top. While the same transmission line with a negative etch factor are 2mm wide on the top and narrower at the bottom. An etch factor of 0 means no etch, and the top and bottom dimensions are the same. Also, very large etch factors — both positive and negative — result in little or no etch.

In HFSS, only signal layers can have an etch factor, dielectric and negative signal layers do not. HFSS supports an etch ratio that is referred to as the **HFSS Etch Factor**, and this etch is used when the mesh is created for an HFSS simulation. Note, however, the etch is not applied to geometry drawings in the 3D Layout view.

For more information see the Edit Layers window.

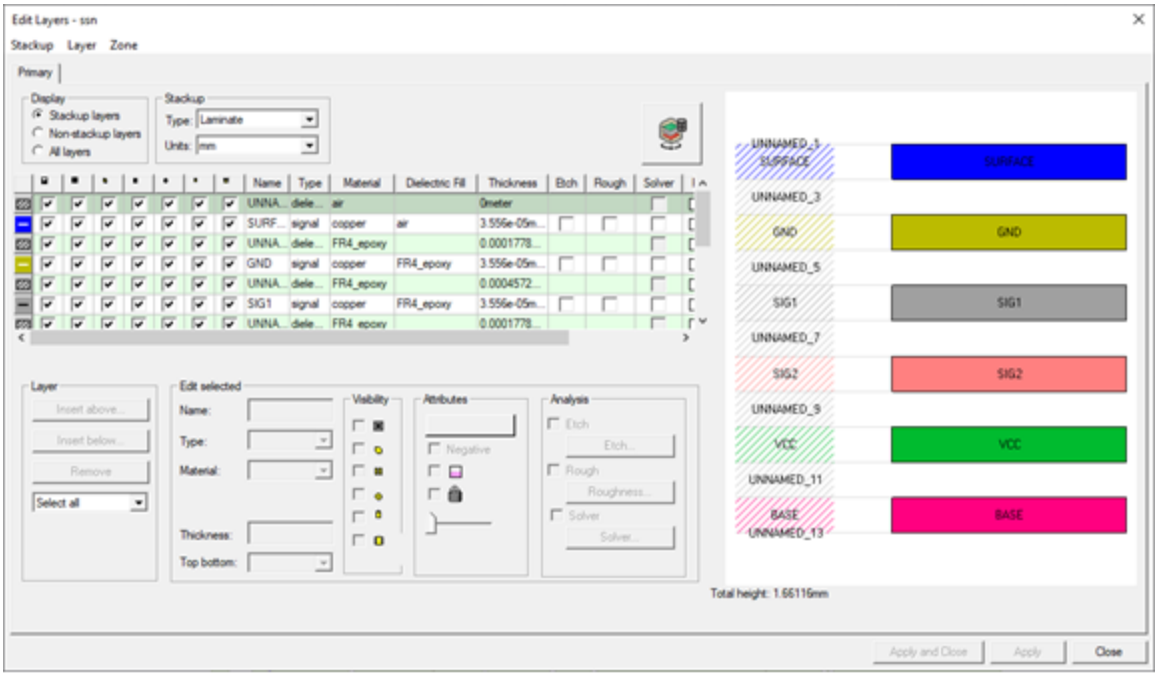
Layout Stackup

A stackup is the list of physical layers in a circuit board. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform offers three types of stackups:

- Laminate: All layers physically stack one another with no overlap on the Z-axis. This is the default type.
- Multizone: Layers physically stack on one another in the Z-axis but each layer can have zones made of different or no material.
- Overlapping: Layers can physically intersect one another on the Z-axis.

A stackup is defined in the Edit Layers window which you can use to add, configure, and remove layers on the stackup. To access this window on the **Layout Editor**, select **Layout** > **Layers**. If you are working in the **Schematic Editor**, select **Schematic** > **Layout Stackup**.

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



Configuring a Stackup

After adding a design to a project, the next step is to create the design's dielectric and signal layers using the **Layout Editor**. Layers are defined and modified in the **Edit Layers** window; use the window to import a stackup defined in an XML file. The XML file that defines the stackup is typically created using a third party text editor such as Microsoft Notepad or VI on Linux. For more information see Importing a Stackup.

Embedded Dielectric Layers

A signal layer can be assigned a dielectric material. And since signal layers are permitted to overlap other layers, create a stackup with dielectric materials embedded within other layers.

HFSS Analysis

Meshes correct modeling, and the dielectric materials are sent to the HFSS solver.

To configure a stackup:

Click anywhere on the **Layout** view window to activate it.

From the **Layout** menu, select **Layers** to open the **Edit Layers** window.

3. Use the window to modify the stackup. Functionality available includes:
   - Import a new stackup from an xml control file.
   - Add a layer.
   - Remove a layer.

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

- Specify whether a layer in the project is editable or visible. Select the color, pattern, and position of each layer in the project. Click on column headings to configure which columns to display.
- Specify the name, material attributes, thickness, elevation, and roughness of each layer in the project. Click on column headings to configure which columns to display.

> **Note:**
>
> Click **Apply** in the Edit Layers window to apply the changes you make; click **Close** to close the window.

Laminate Stackup

In a laminate stackup, all layers physically stack one another with no overlap on the Z-axis. Laminate layers extend over the entire design space in the X-axis and Y-axis. Laminate stackups are the default type of stackup.

With a laminate stackups, all controls appear on a single tab of the **Edit Layers** window. Options are available in the **Stackup** and **Layers** menus but not the **Zone** menu. When a new layer is created, the window automatically offers the opposite type of layer from what is highlighted in the **Grid Control** pane (e.g., a new signal layer if a dielectric layer is highlighted).

The lower and upper elevation columns are not visible by default but can be activated by the user. The thickness of layers may be edited but not the lower elevation. Layers may be reordered by dragging a layer to a new position in the **Grid Control**. See Elevation in Laminate and Multizone Stackups.

When visible, a laminate stackup layer has its **Dielectric Fill** specified for signal layers.

Multizone Stackups and Rigid-Flex Bends

Creating a multizone stackup allows a design to be divided into areas called zones. Each zone contains a subset of layers available to the entire circuit board and is spatially defined by polygons on the **Outline** layer. When an object (e.g., a primitive) is placed on a layer, it is also placed in a zone. Objects belong to the zone where their center point resides, even if part of the object is in another zone. Likewise, terminals and ports also belong to the zone in which they reside. If an object with terminals is placed over a zone boundary, the terminals are automatically assigned to the zone they are in. Those terminals may only be placed on the layers available in their zone. If a zone boundary changes, the zone placements of all terminals, padstacks, and objects are automatically recomputed. Objects in a zone may only be placed on layers in that zone (e.g., if a voltage source is placed over multiple zones, its individual terminals are assigned to the zones in which they are located).

Adding rigid-flex bends to circuit board designs allows users to reshape the design geometry to simulate realistic mechanical behavior and electrical connectivity.

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 cannot remove a layer from a zone if there are objects on that layer and in that zone.

Follow the steps in each subsection to define a multizone stackup and add rigid-flex bends to a design model.

Continue to Creating a Multizone Stackup.

Creating or Importing a Multizone Stackup

Follow the steps in either subsection to create or import a multizone stackup.

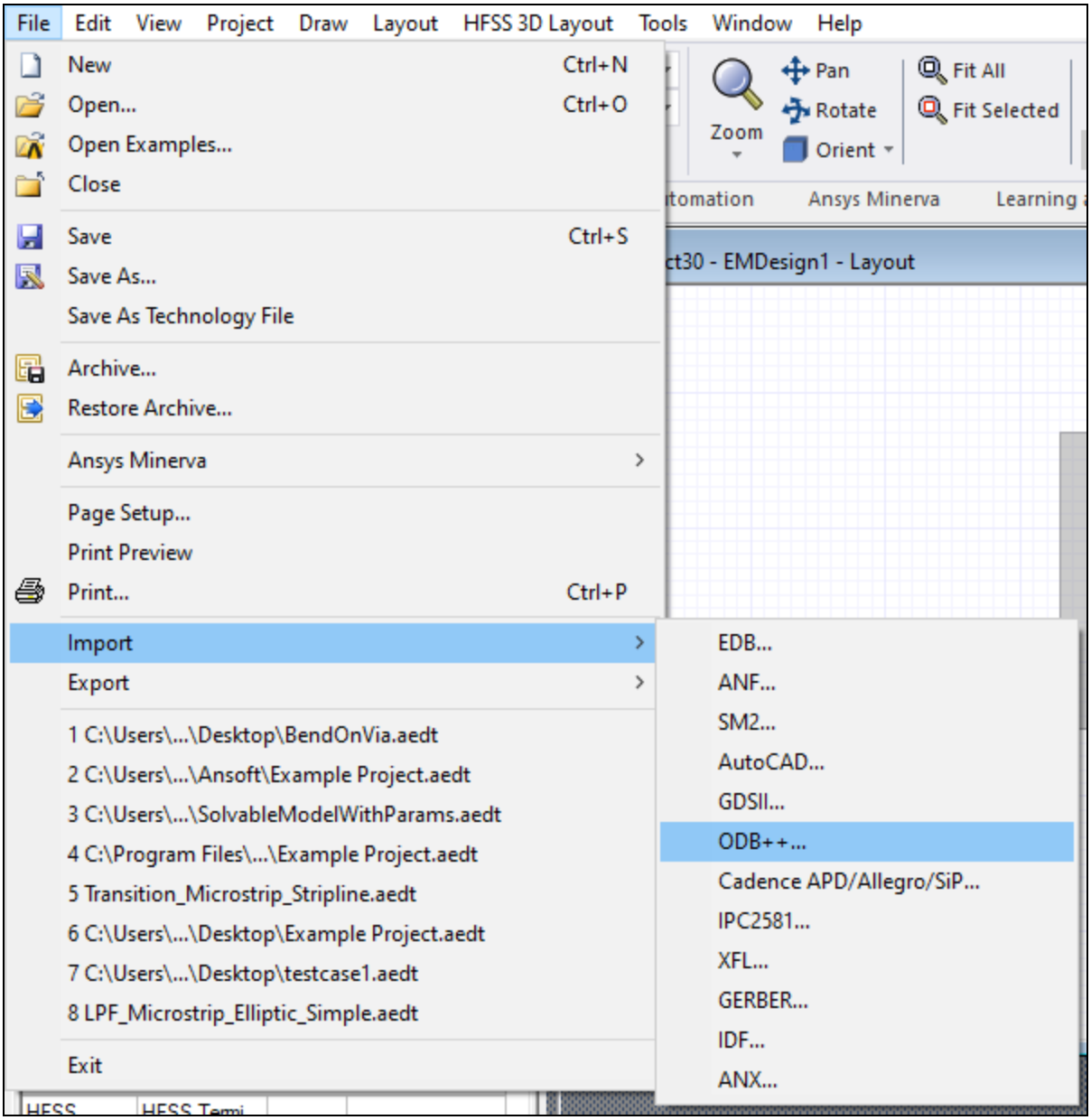
> **Note:**
>
> Multizone stackup import supports ODB++ *.*tgz*, Cadence *.*brd*, and IPC2581 rev. c *.*cvg* or *.xml* file types.

Importing a Multizone Stackup

Follow these steps import a multizone stackup.

1. From **File**, select **Import** > **ODB++**, **Cadence APD/Allegro/SiP**, or **IPC2581** to open an explorer window.

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



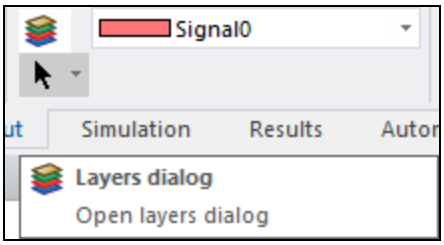
2. Navigate to an appropriate file. Then double-click the file or select the file and click **Open**.
3. If applicable, refer to Importing ODB++ into the Layout Editor, Importing Cadence APD/ALLEGRO/SIP Files into the Layout Editor, or Importing IPC2581 Files into the Layout Editor.
4. Make further changes to the imported stackup by completing the steps in Creating a Multizone Stackup.

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 Multizone Stackup

Follow these steps to enable multizone editing and add zones to a project.

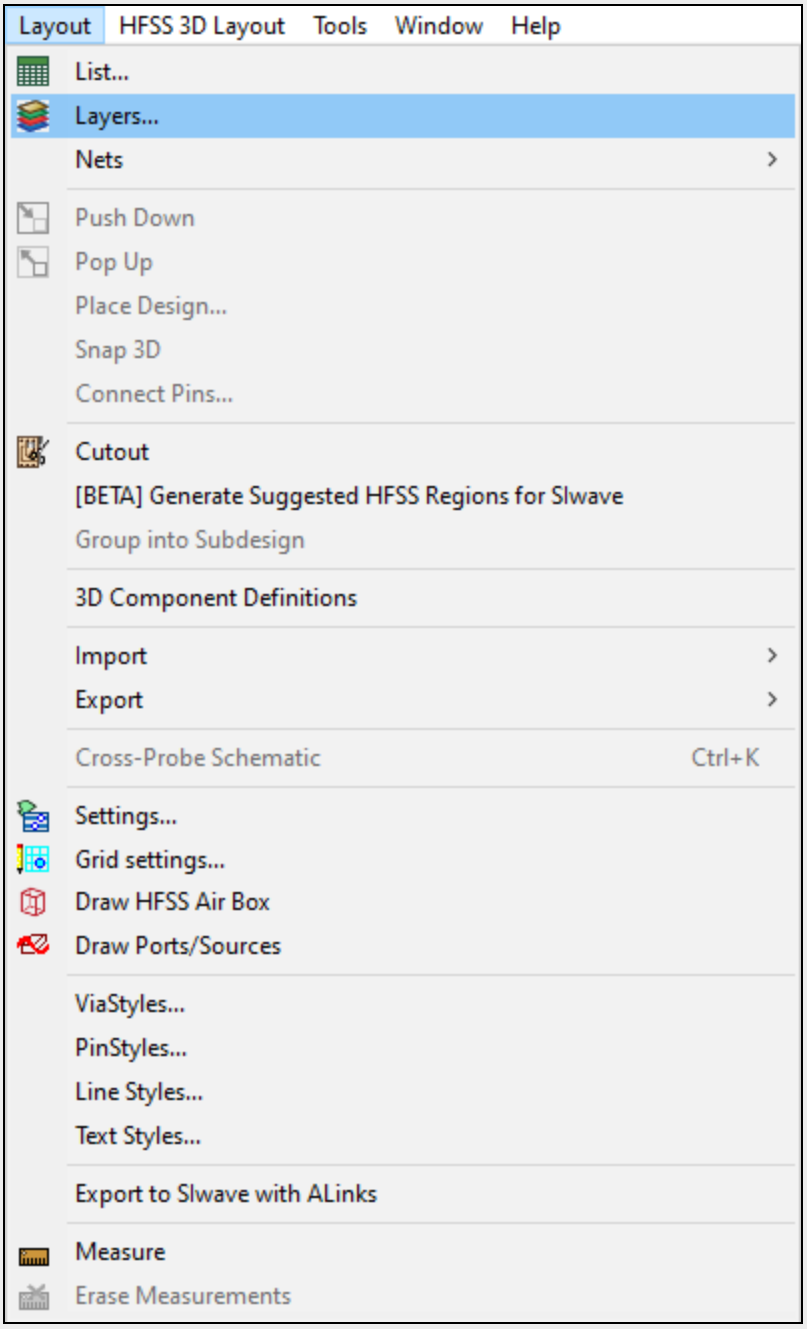
1. Once **Electronic Desktop** restarts, **Create** or **Open** a project. Refer to "Creating Projects" on page 3-77 or "Opening Projects" on page 3-71.
2. From the **Layout** tab, select **Layers dialog** to open the **Edit Layers** window.



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

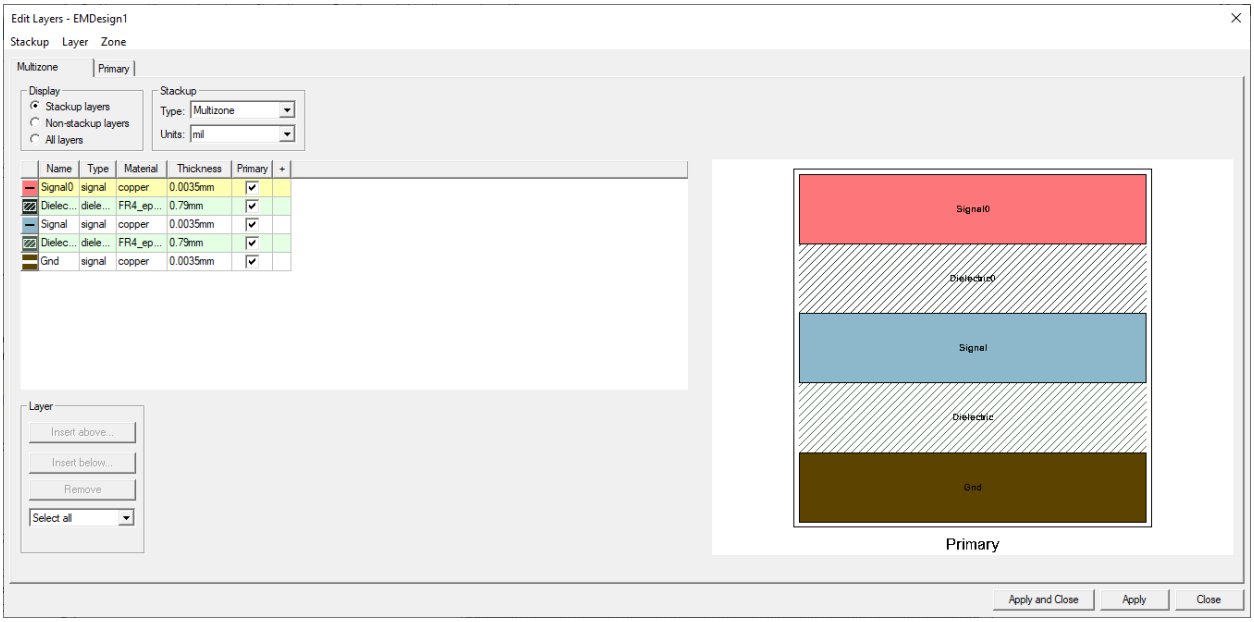
Note:

Alternatively, from **Layout**, select **Layers** to open the **Edit Layers** window.



3. Select **Multizone** on the **Type** drop-down menu and the **Multizone** 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.



4. If the design was imported with multiple zones already active, make any appropriate changes using the same methods described in the The Edit Layers Window for editing layers.

Continue to Creating or Copying Zones.

Creating or Copying Zones

Follow the instructions in the subsections below to create or copy zones.

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

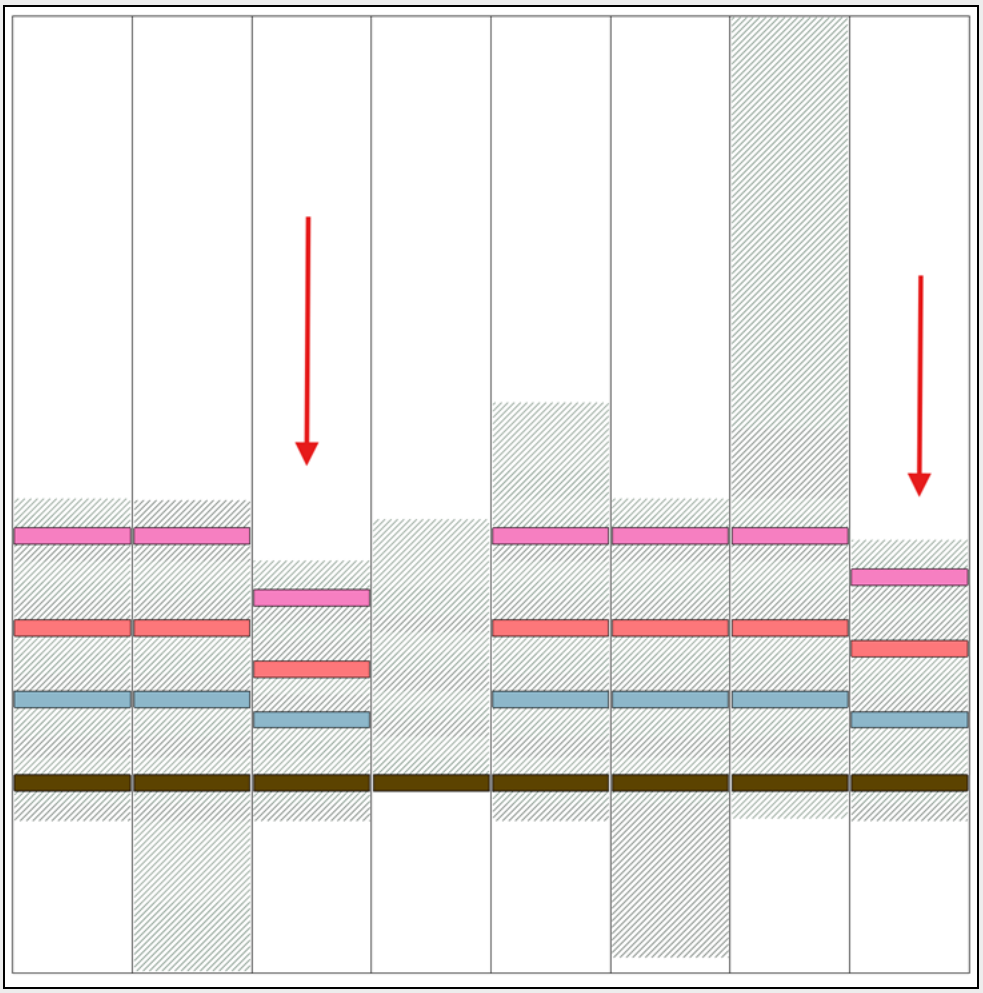
Note:

When creating a multizone stackup, it is important to ensure that signal layers align consistently between zones. Each zone has its own local stackup, which is a subset of the full board layer structure. If a signal layer in one zone does not align vertically with the corresponding signal layer in another zone, a discontinuity is introduced at the boundary between the zones.

For example, if "Signal Layer 1" exists at different vertical positions in Zone 1 and Zone 2, or is absent in one of them, the transition between zones may result in unintended electrical behavior (e.g., an open circuit). There is no automatic correction in pre-processing or meshing to resolve misaligned or discontinuous stackups across zone boundaries. These discontinuities are preserved in the mesh and can impact simulation results.

The following image shows a discontinuous signal layer transition between zones, indicated by red arrows.

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 New Zone

From the **Edit Layers** window, click **+** adjacent to the right-most column header to add a new zone with the same layers. Repeat as selected.

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

Note:

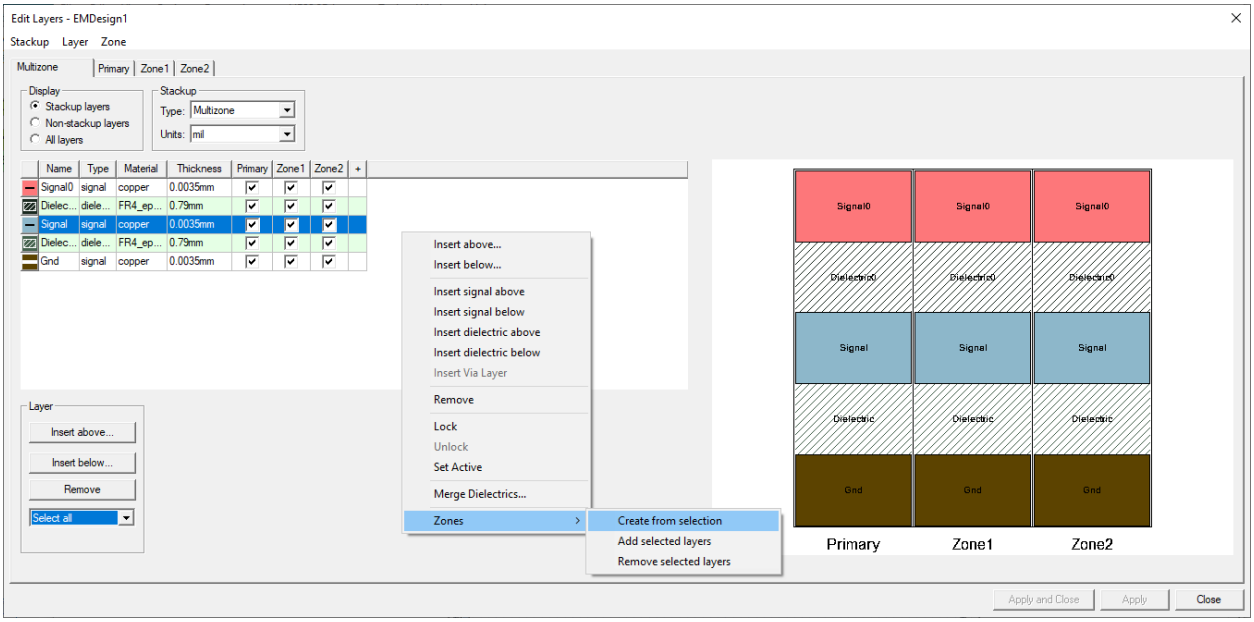
Zones will be added in sequence (i.e., to **Primary**, which appears by default and includes all undefined zones, will be added **Zone 1**, **Zone 2**, et cetera). A new tab appears for each zone:

- **Multizone** – lists all layers and allows users to allocate layers to zones.
- **Primary** – controls layers in the **Primary** zone. The **Primary** zone may contain a subset of layers, but all components not contained in a **Zone** are treated as if they are in the **Primary** zone.
- **Zone*N*** – control layers in their respective zone. Each zone is labeled with the corresponding zone's designation.

Adding a New Zone With Specific Layers

From the **Edit Layers** window, **Ctrl**+**click** anywhere within the selected layer(s). Then either right-click and select **Zones** > **Create from selection** or navigate to **Zones** > **Create from selection**.

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



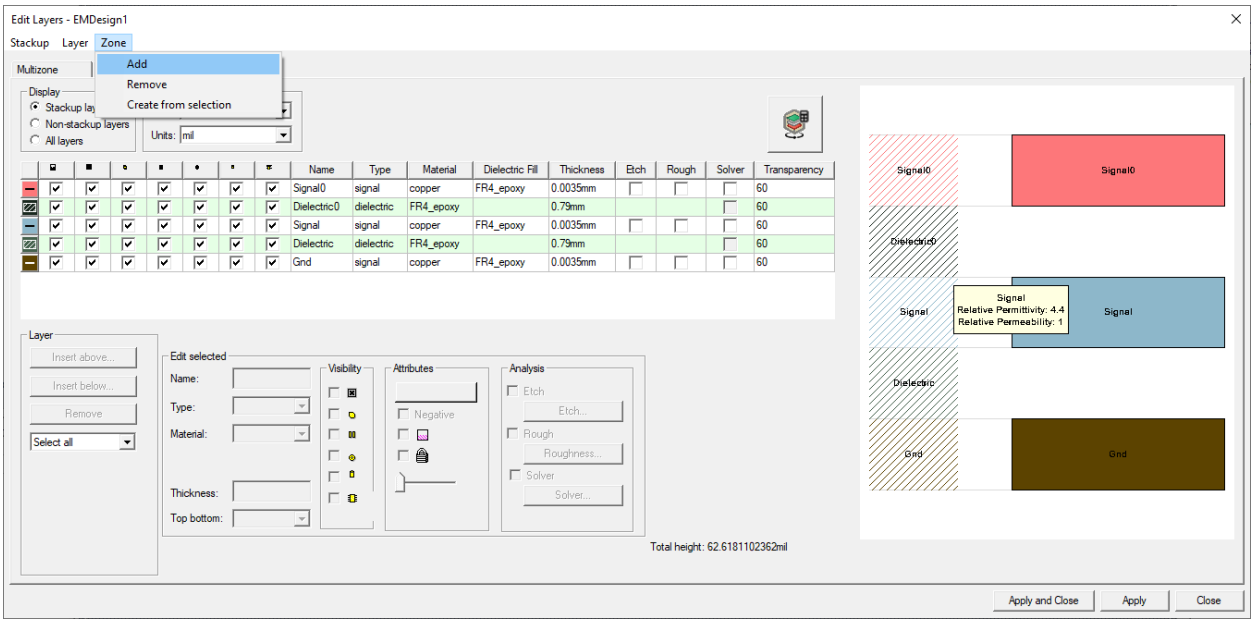
The new zone has only the highlighted layer(s).

Copying a Zone

From the **Edit Layers** window, navigate to the selected zone's tab and make any appropriate changes. Then navigate to **Zone** > **Add** to copy the selected zone to the next sequential number

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

(e.g., copying **Zone 2**, when **Zone 3** has already been added, will designate the copy as **Zone 4**).



Continue to Editing Zones.

Editing Zones

Once zones are created, follow these steps to edit the zones.

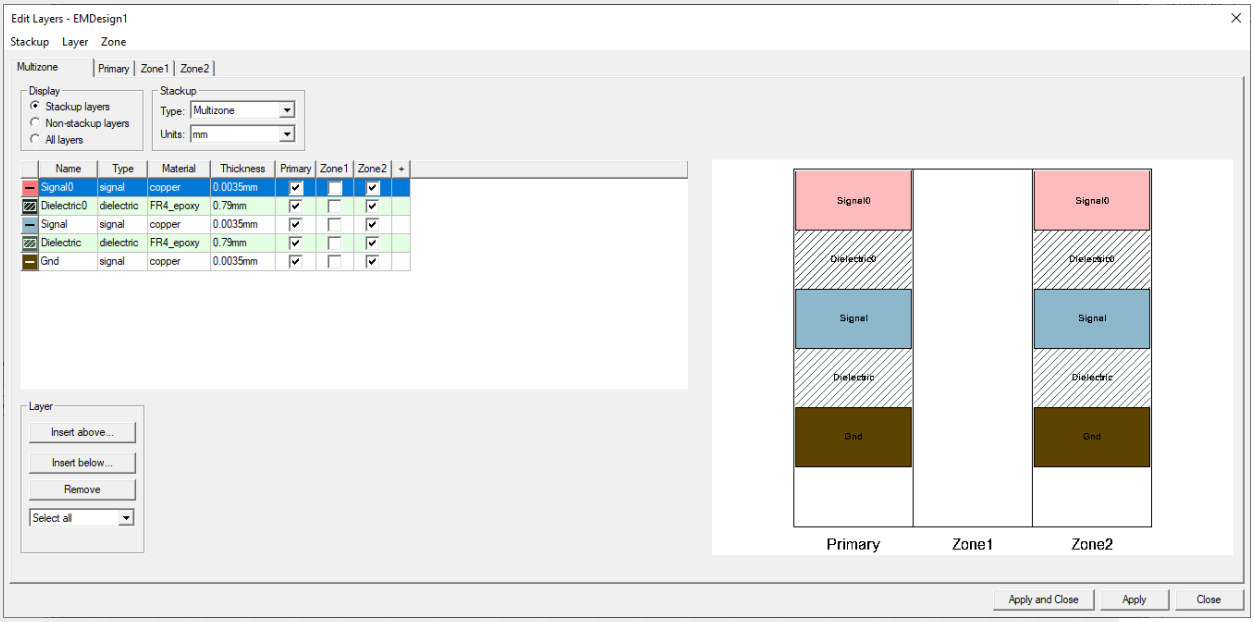
1. Edit the new zones (e.g., add or remove layers, et cetera) using the same methods described in the The Edit Layers Window for editing layers.

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

Note:

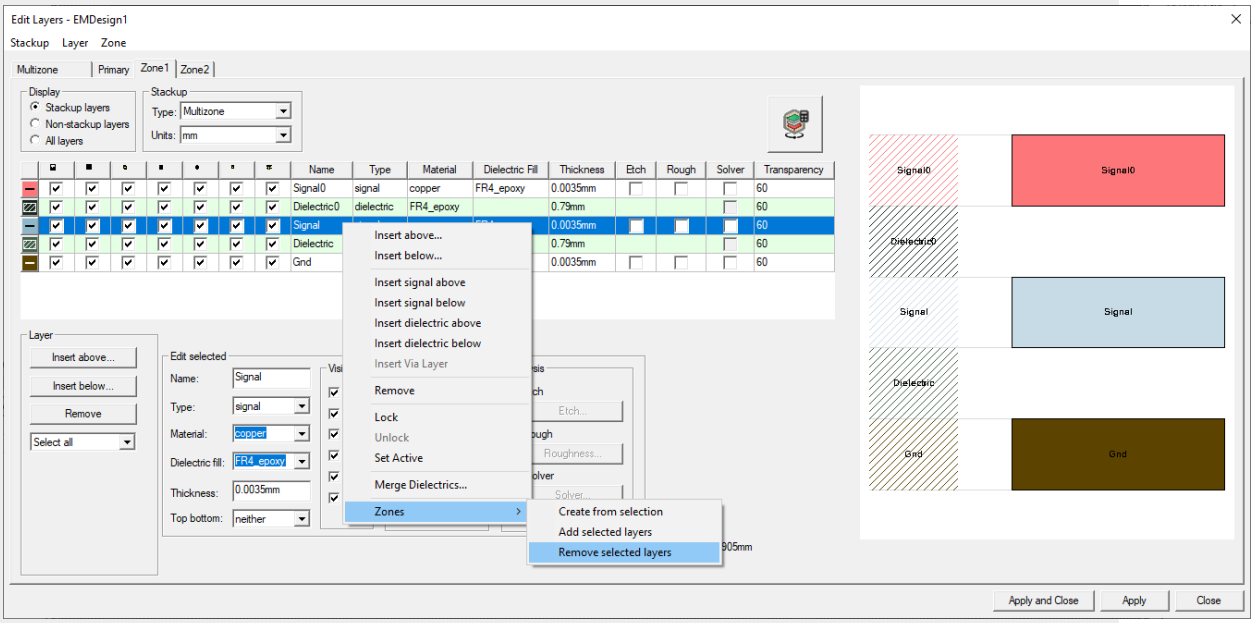
If a layer cannot be removed, check that an object (e.g., a primitive) is not still on that layer or in that zone. Then do one of the following:

- Remove the check on the box in the zone's column. View the change in the preview window.

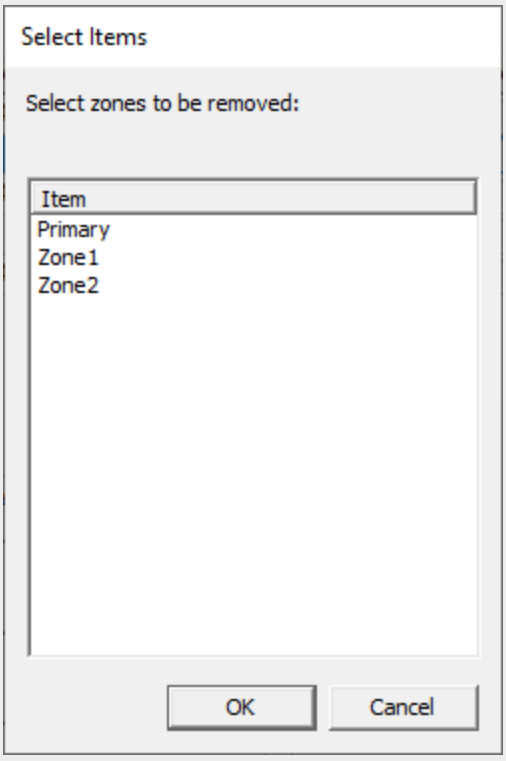


- Select the inappropriate layer or **Ctrl+click** to select multiple layers. Then right-click and select **Zones** > **Remove selected layers** to open the **Select Item** window.

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 appropriate zone and click **OK** to close 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.

Note:

Layers can also be removed from zones after bends have been created and defined (i.e., after Defining Rigid-Flex Bends).

2. Click **Apply and Close** to close the **Edit Layers** window and return to the **Layout Editor**.

Continue to Defining Zones.

Defining Zones

Follow these steps to outline and define the new zone(s) using the **Rigid-Flex Workflow**.

Note:

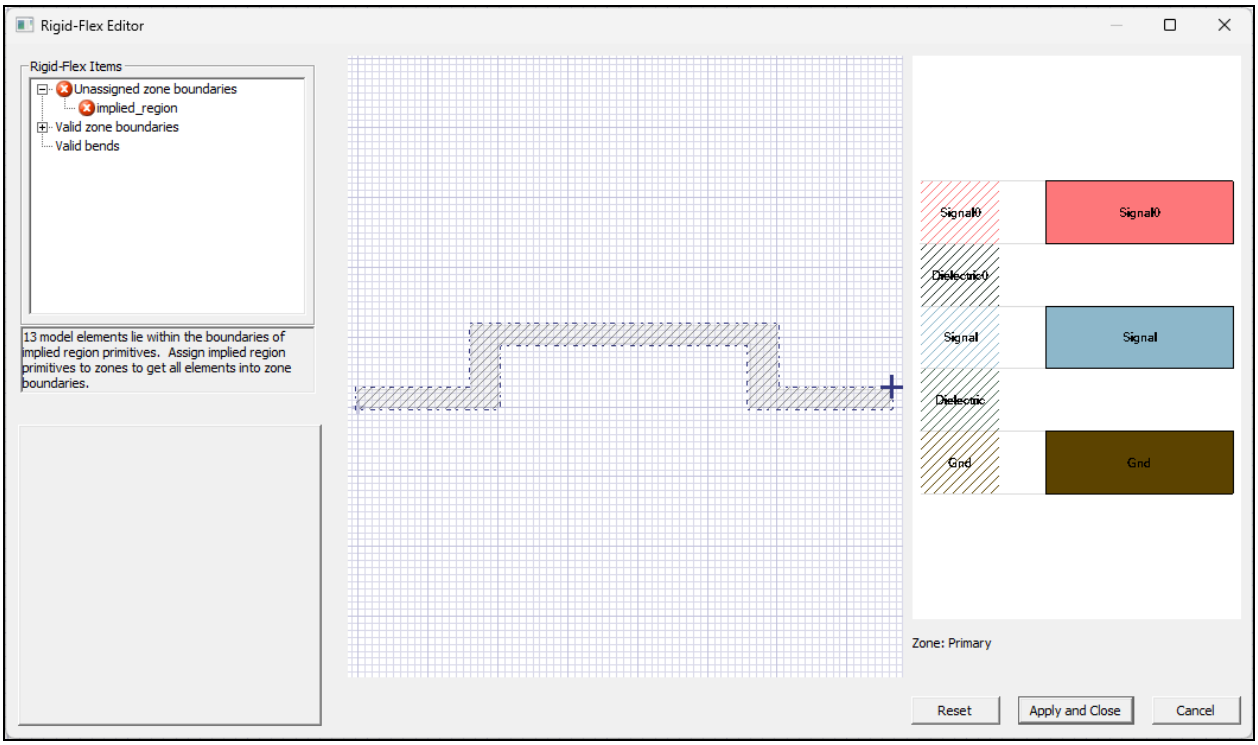
The **Rigid-Flex Workflow** is only available after a design has been imported or created, and **Multizone** is selected as the stackup's **Type**. Refer to the previous subsections of the "Multizone Stackups and Rigid-Flex Bends" on page 6-31.

Note:

Zones can be defined at the **Layout Editor**, but the **Rigid-Flex Editor** window provides a dedicated environment for managing rigid-flex zones and bends, and includes a specialized message window that displays warnings and setup-related messages not shown in the standard **Message** window. These additional messages can help identify potential configuration issues early in the design process, making the **Editor** a more robust option for preparing complex rigid-flex models.

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

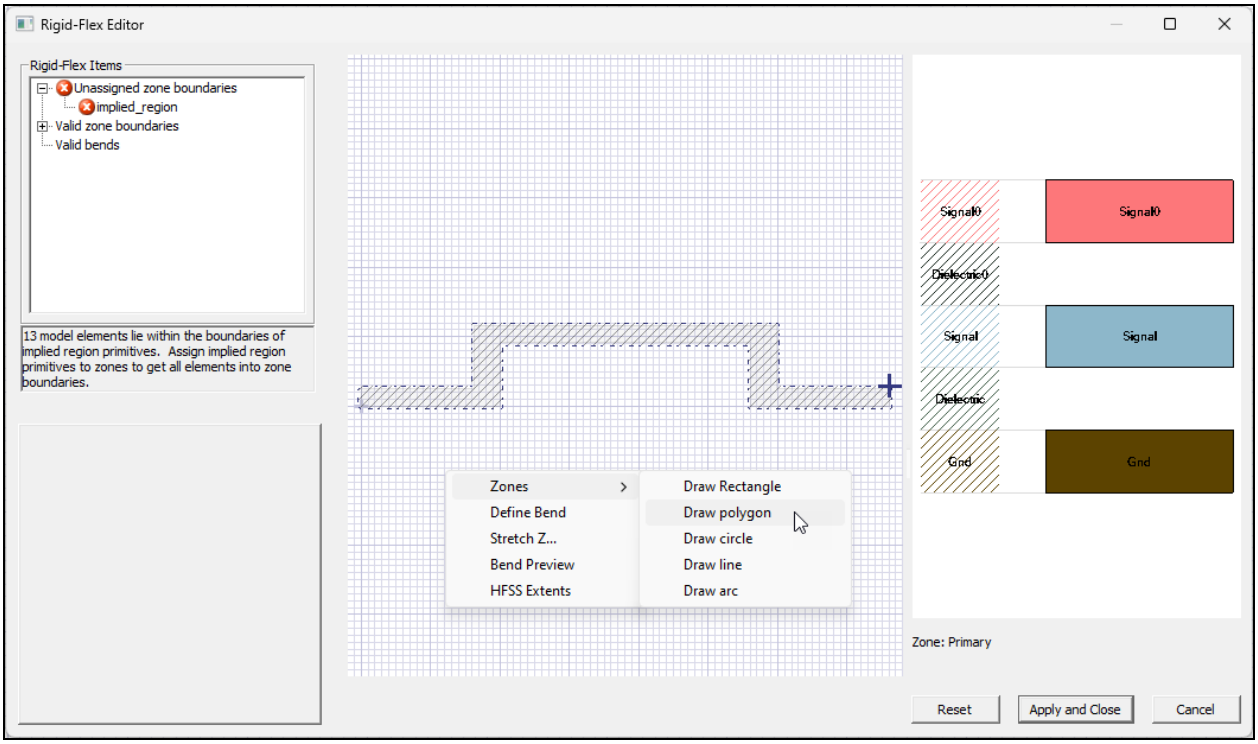
1. From the **Layout** tab, click **Rigid-Flex** to open the **Rigid-Flex Editor** window.



> **Note:**
>
> As zones are created, they will appear in the **Rigid-Flex Items** area. Outline primitives that are not assigned to a zone will be located in the **Unassigned zone boundaries** section as **implied region(s)**.

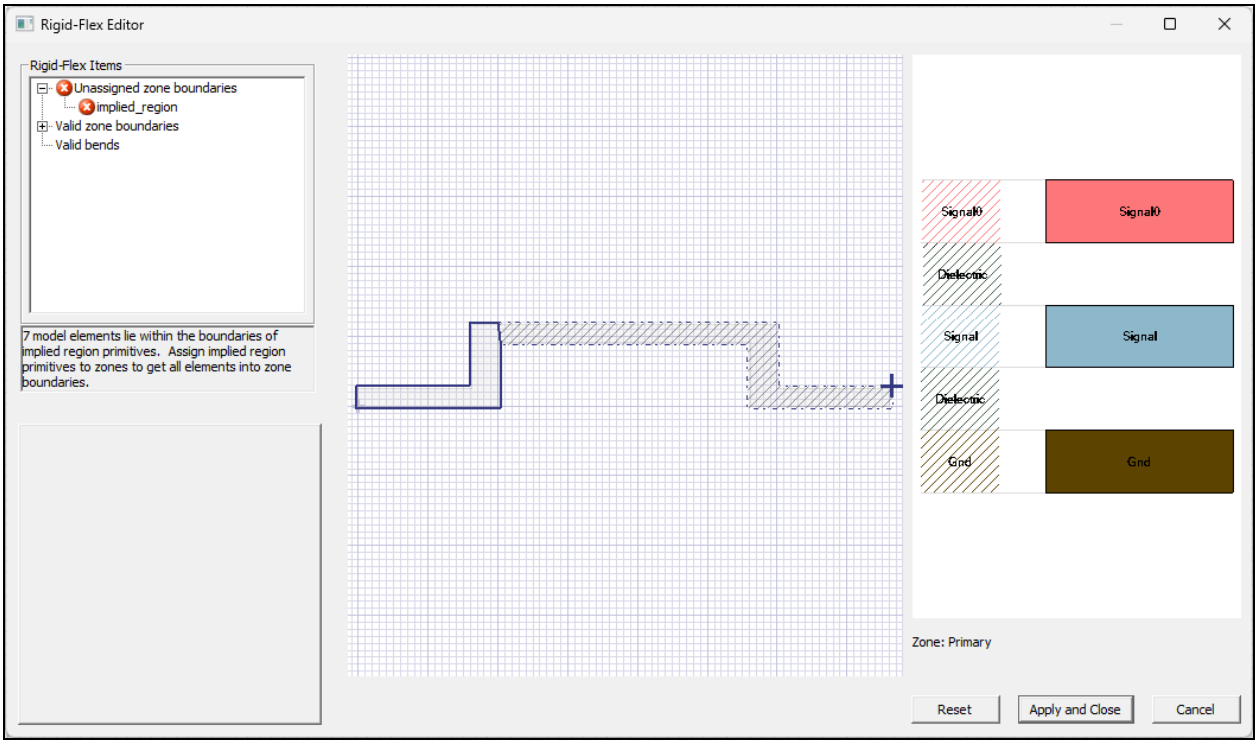
2. From the **Rigid-Flex Editor** window, define a zone by doing the following:
   a. Right-click within the preview window and select **Zones** > [*any primitive drawing tool*] (e.g., **Draw polygon**).

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



b. Click within the preview window to draw a zone using the same method described in the HFSS 3D Layout Getting Started Guides for drawing primitives in the **Layout Editor**.

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

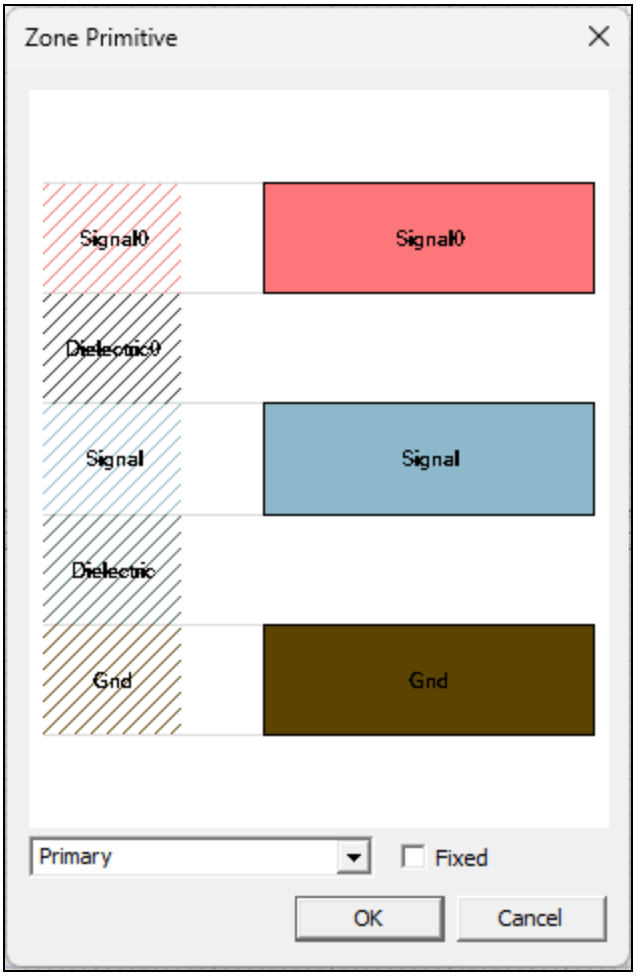


Note:

Click **Reset** at anytime to undo any changes and begin again.

c. Double-click to complete the outline and open the **Zone Primitive** window.

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



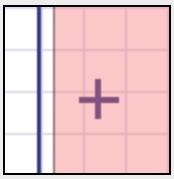
d. From the **Zone Primitive** window, select the appropriate zone designation from the drop-down menu (e.g., **Primary**).

e. Check the **Fixed** box, if applicable. A fixed primitive is used as reference during the bending process and contains the anchor point.

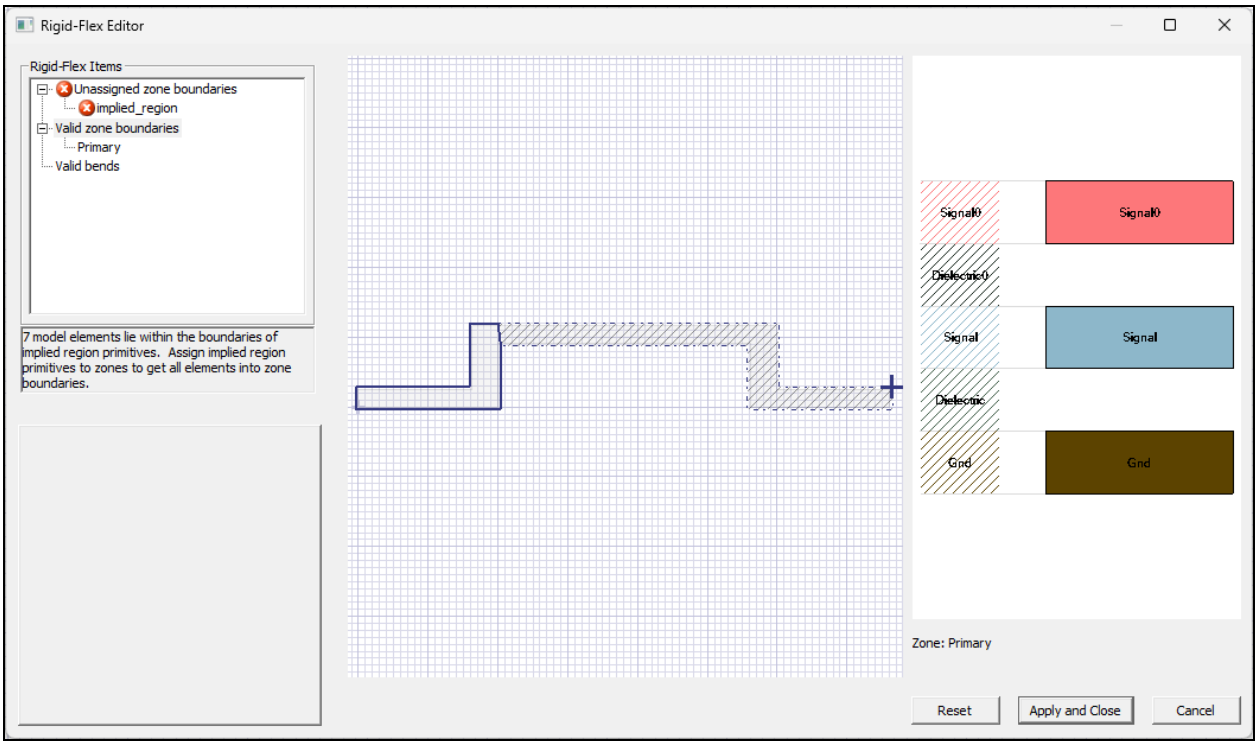
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The anchor point appears as a bold crossbar within the design. Its position will remain stationary during the bending process. Although a bend can be applied to the primitive that contains the anchor point, it cannot intersect the anchor point. The anchor point can be dragged to another location or even another primitive. Any primitive that contains the anchor point is **Fixed**. The anchor point is visible in the **Layout Editor** if the user is Defining Zones.



f. Click **OK** to close the **Zone Primitive** window.



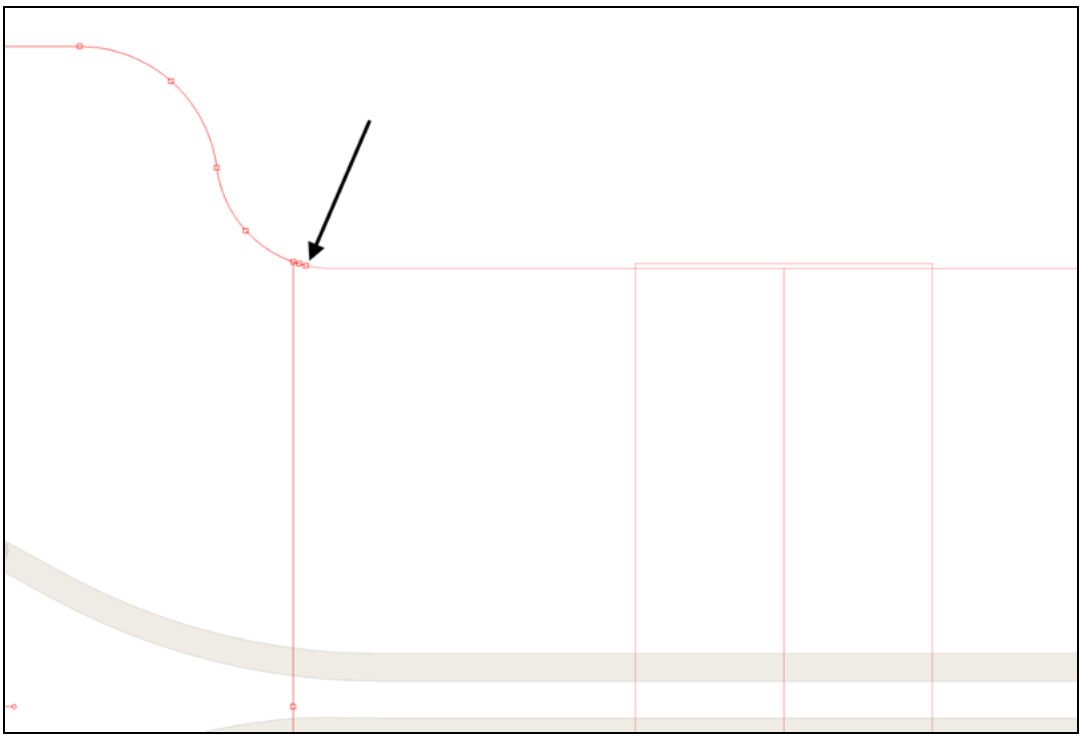
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Understanding Zone Boundaries

Although simulation includes a robust pre-processing routine to automatically clean up zone boundary definitions, *starting* with clean, well-defined zones greatly improves simulation reliability and performance. Rather than relying on implied regions, ensure proper behavior during simulation and meshing by verifying that zone boundaries are clearly defined on the **Outline** layer. Ensure the following:

- Zones are explicitly defined using polygons and no implied regions remain when the design is ready for simulation.
- Zone polygons are well-formed and do not contain gaps between adjacent regions.
- **No** components cross defined zones.
- Zone boundaries do not create zero-area regions.

The following image shows a zero-area region that could potentially cause meshing errors.

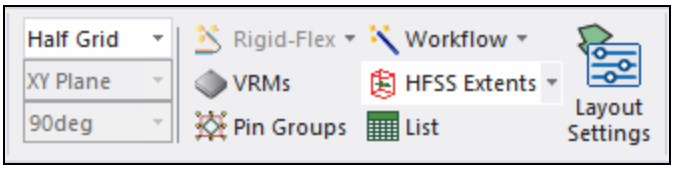


Using HFSS Model Extents to Modify Zones

Follow these steps to use **HFSS Extents** to modify the dielectric and airbox extents. Refer to HFSS 3D Model Extents.

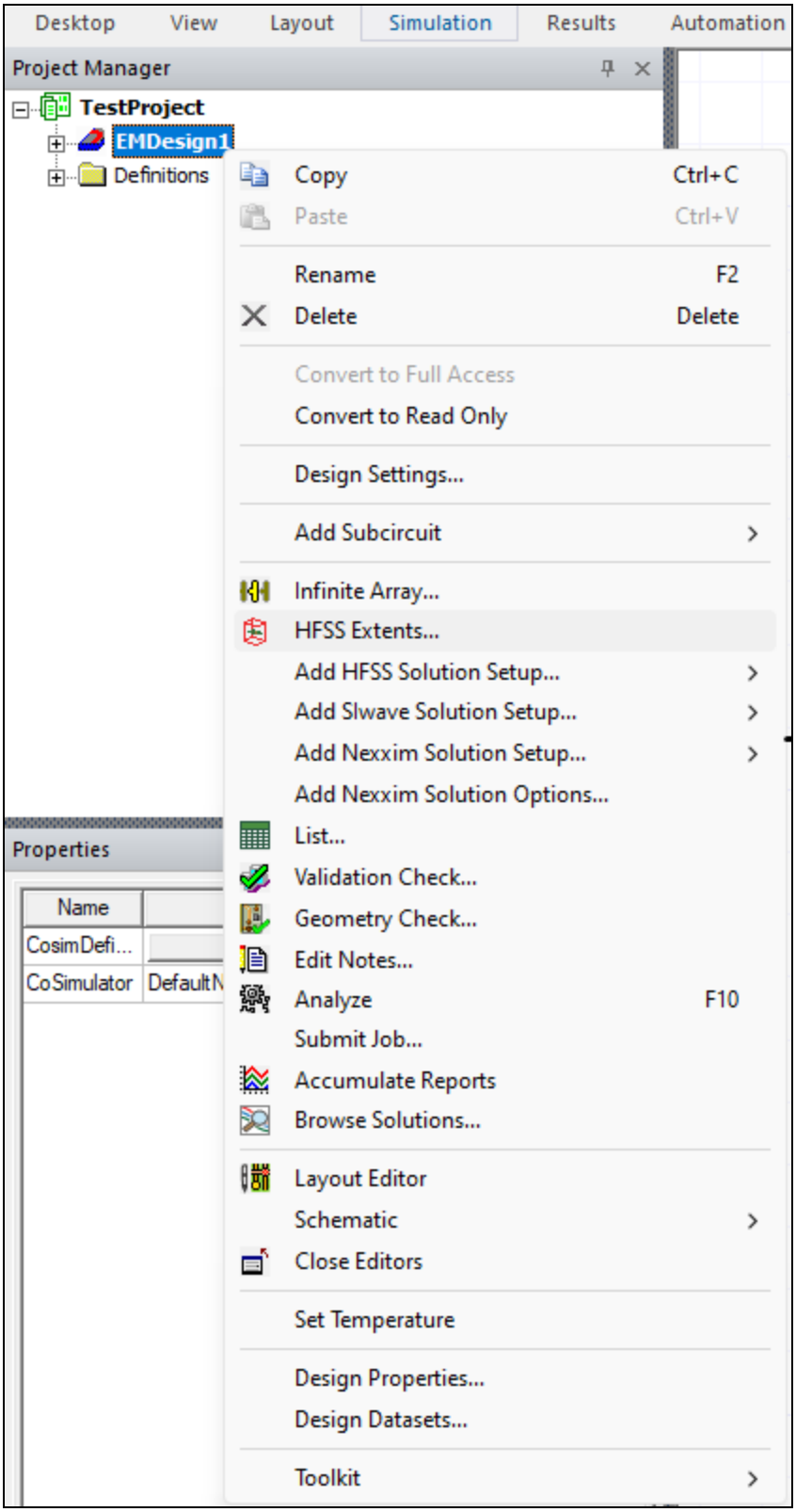
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **Set HFSS Model Extents** window, do either of the following:
   - From the **Layout** ribbon, click **HFSS Extents**.



   - From the **Project Manager** window, expand the **Project Tree** (e.g., **TestProject**). Then right-click the active design (e.g., **EMDesign1**) and select **HFSS Extents**.

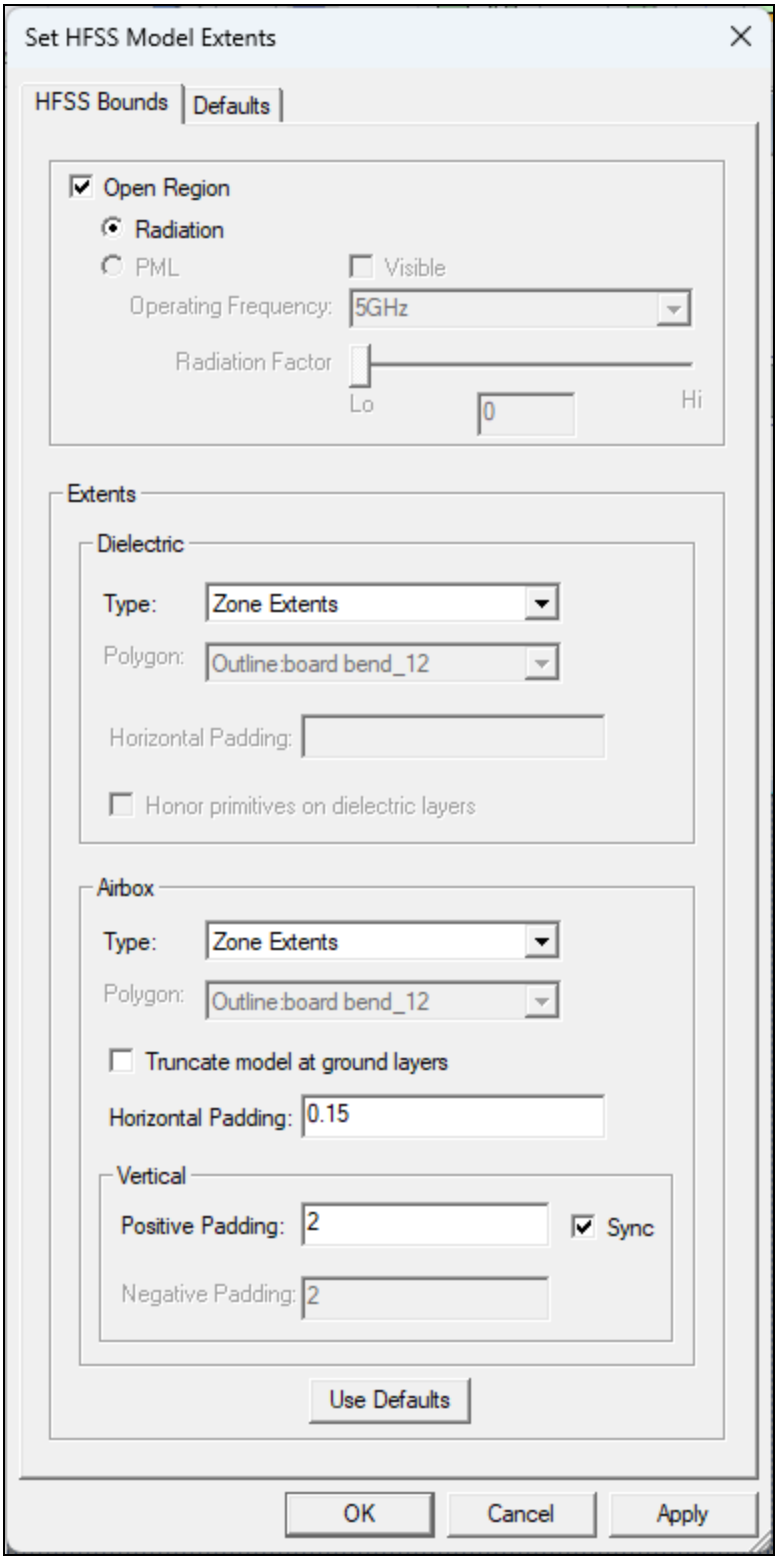
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

The **HFSS Bounds** tab displays 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.



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

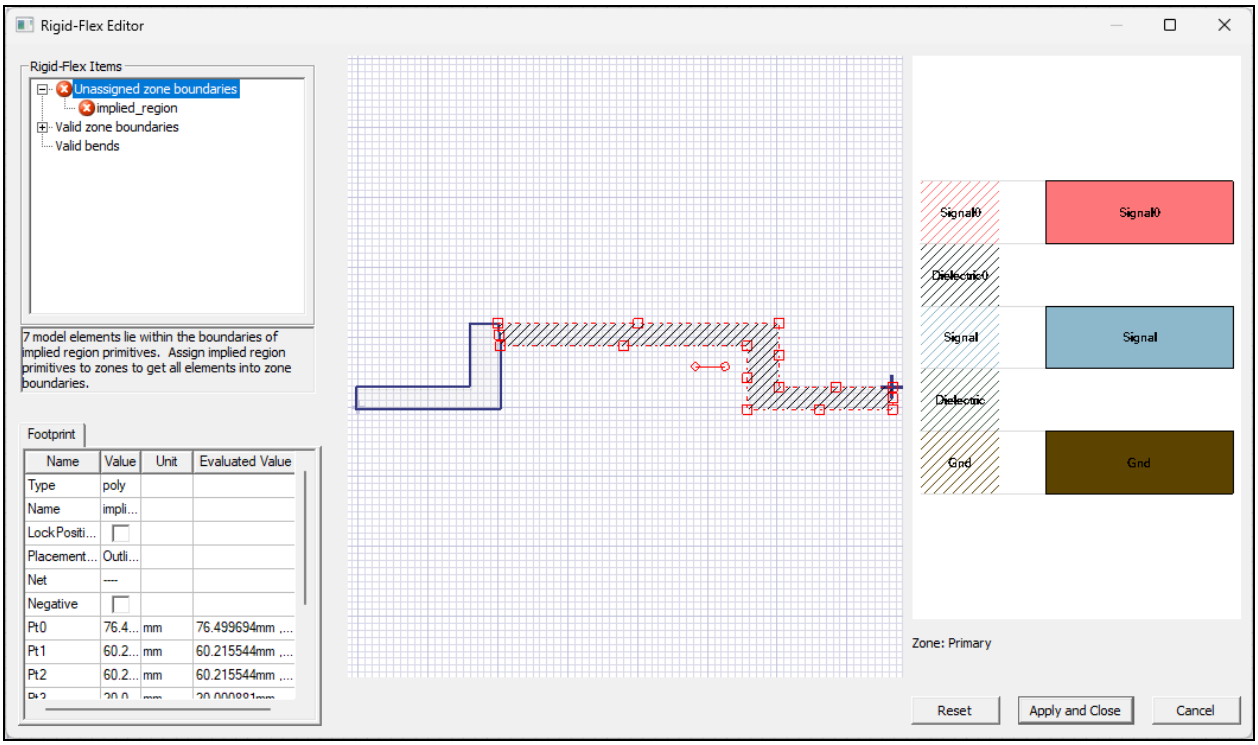
2. From the **Extents** area, select **Zone Extents** from the **Type** drop-down menu to affect the following changes:

- **Dielectric** – the base-polygon is defined by the exact dimensions of the zone extents input by the user on the **Outline** layer.
- **Airbox** – creates a union from the zone extent polygons on the **Outline** layer, expanded by the user-entered value in the **Horizontal Padding** field, to set the airbox extents.

> **Note:**
>
> Padding applied using **Zone Extents** affects **only** implied regions. It has no effect on zones with explicitly defined boundaries. For higher accuracy and more predictable modeling, always use explicit zone boundaries in conjunction with **Zone Extents**.

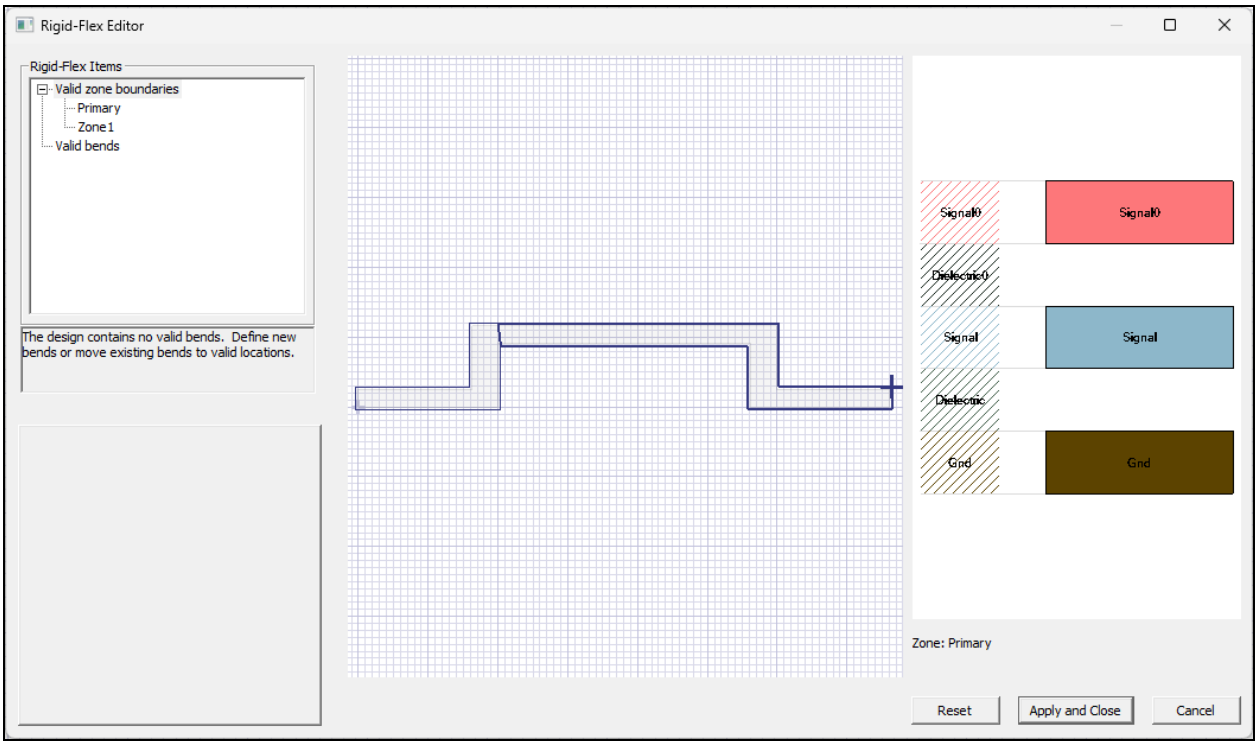
Using the Implied Regions to Assign Zones

The **Unassigned zone boundaries** section of the **Rigid-Flex Items** area includes all primitives that are not assigned to a zone. The primitives that *can* be assigned to a zone are **implied_region(s)**.



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 any **implied_region** to open the **Zone Primitive** window. Then assign the primitive to a zone (e.g., **Zone1**).



Note:

Depending on whether the unassigned primitives are connected, there could be multiple **implied_regions** in the **Unassigned zone boundaries** section.

Changing the Zone Order

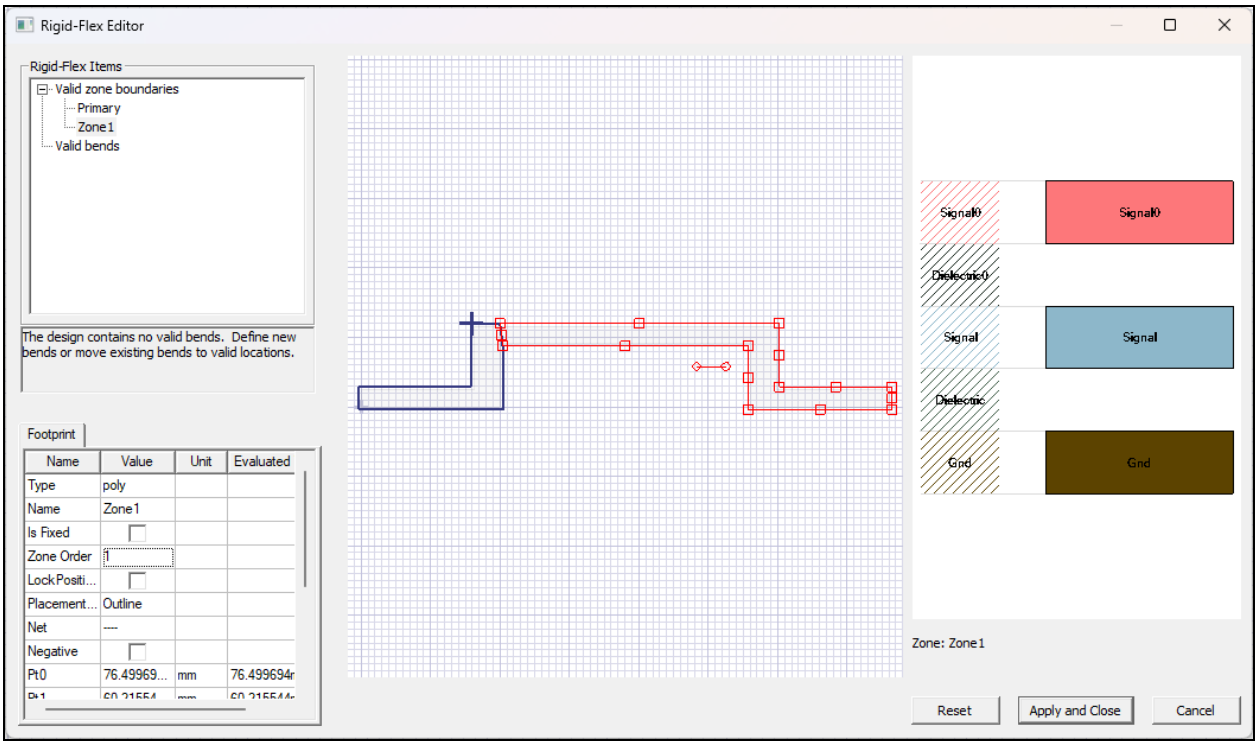
While creating a rigid-flex design, only a single zone stackup will be modeled in a location. If zones overlap, **Zone Order** is used to determine which zone the design will model and which zone(s) will be ignored. The zone with the most dominant position will take precedence. Follow these steps to change the **Zone Order** of any zone in the design.

Note:

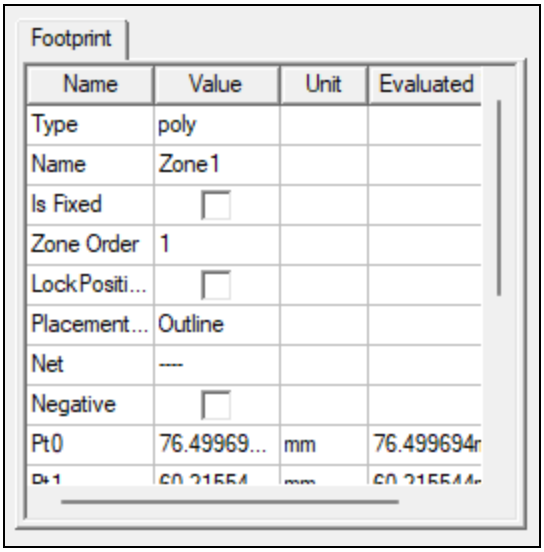
Zone Order can an also be changed at the **Layout Editor**.

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 **Rigid-Flex Editor** window, select a zone.

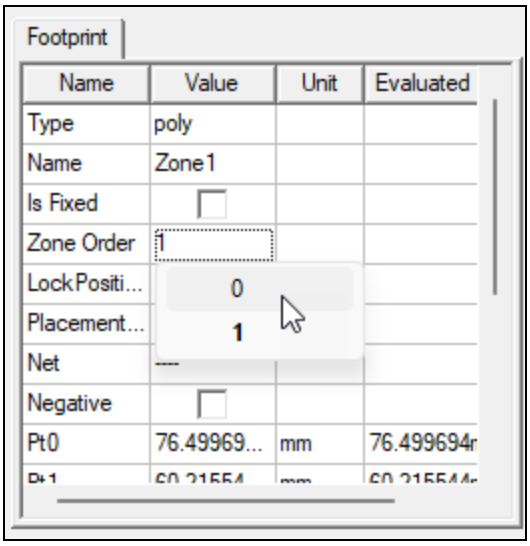


The selected zone's parameters populate the **Footprint** tab.



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

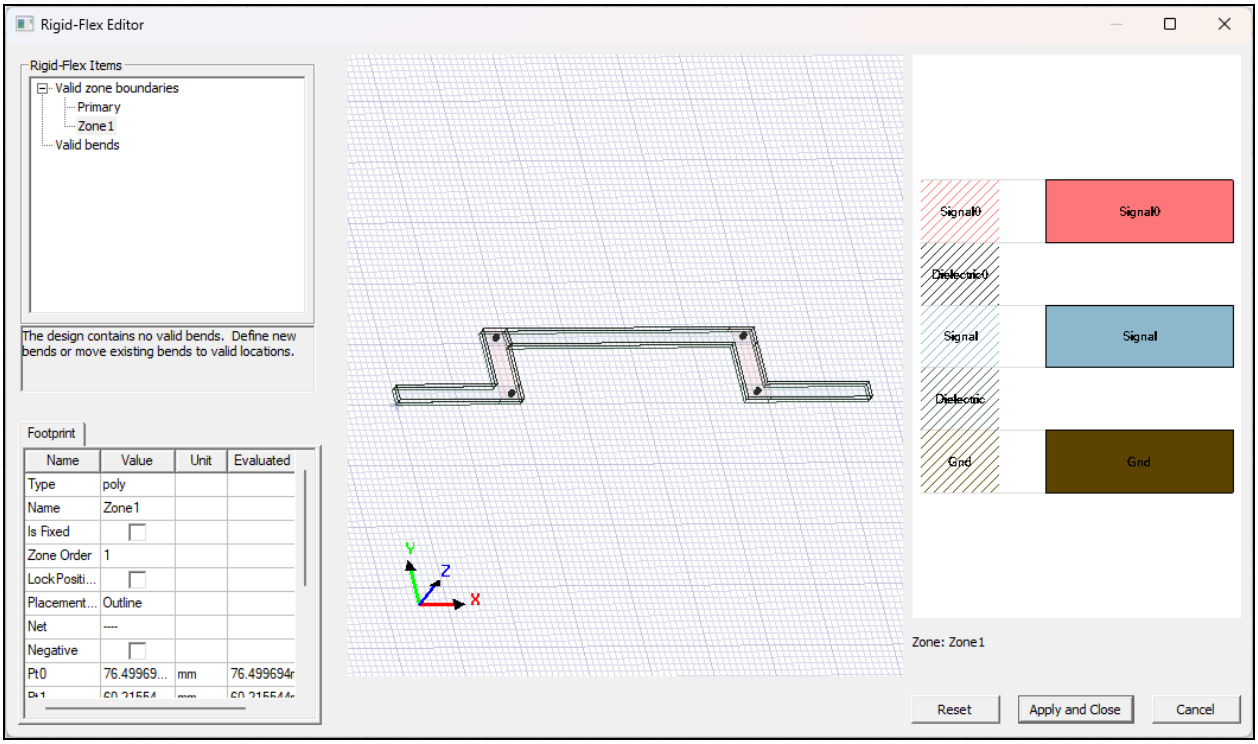
2. From the **Footprint** tab, click inside the **Zone Order** field to select a new position from the drop-down menu (e.g., from **0**, which has the highest priority in the hierarchy, to **1**, **2**, et cetera).



Reorienting the Model and Stretching Z

Reorient the model in the preview window using the same method of control common to the **Layout Editor**.

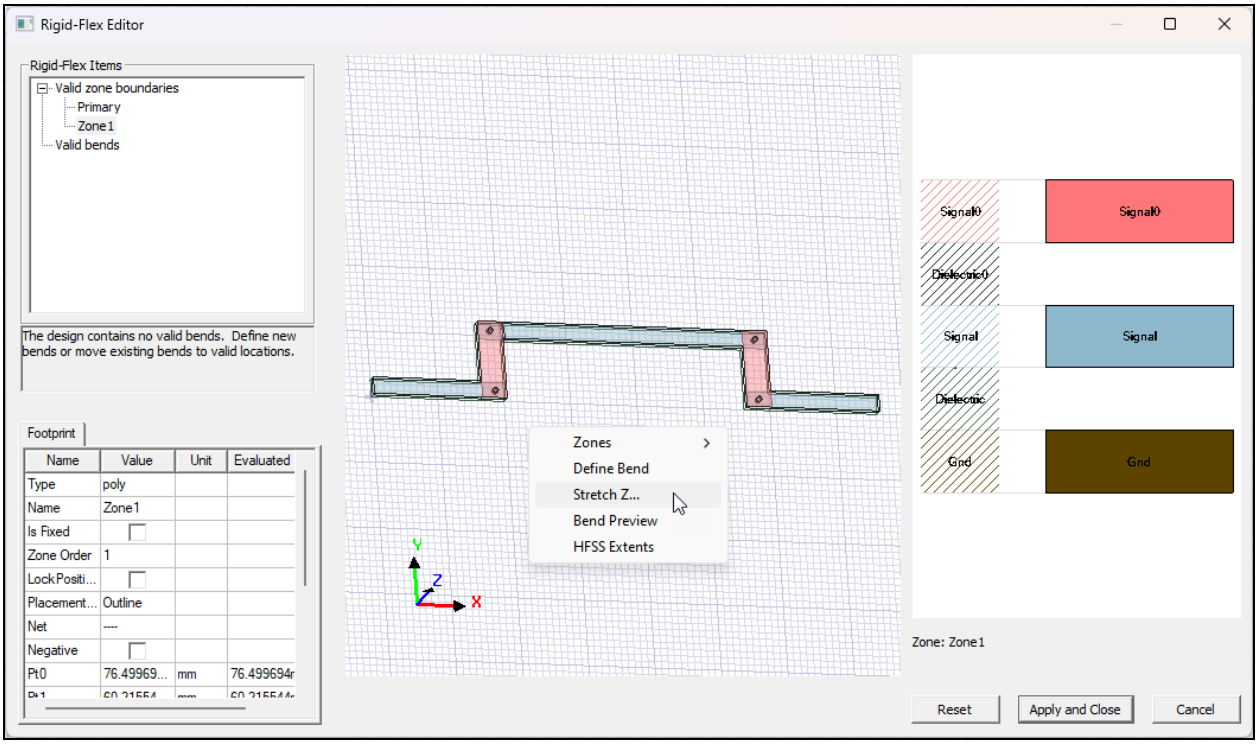
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If applicable, follow these steps to artificially scale Z values, so thin models have a 3D appearance.

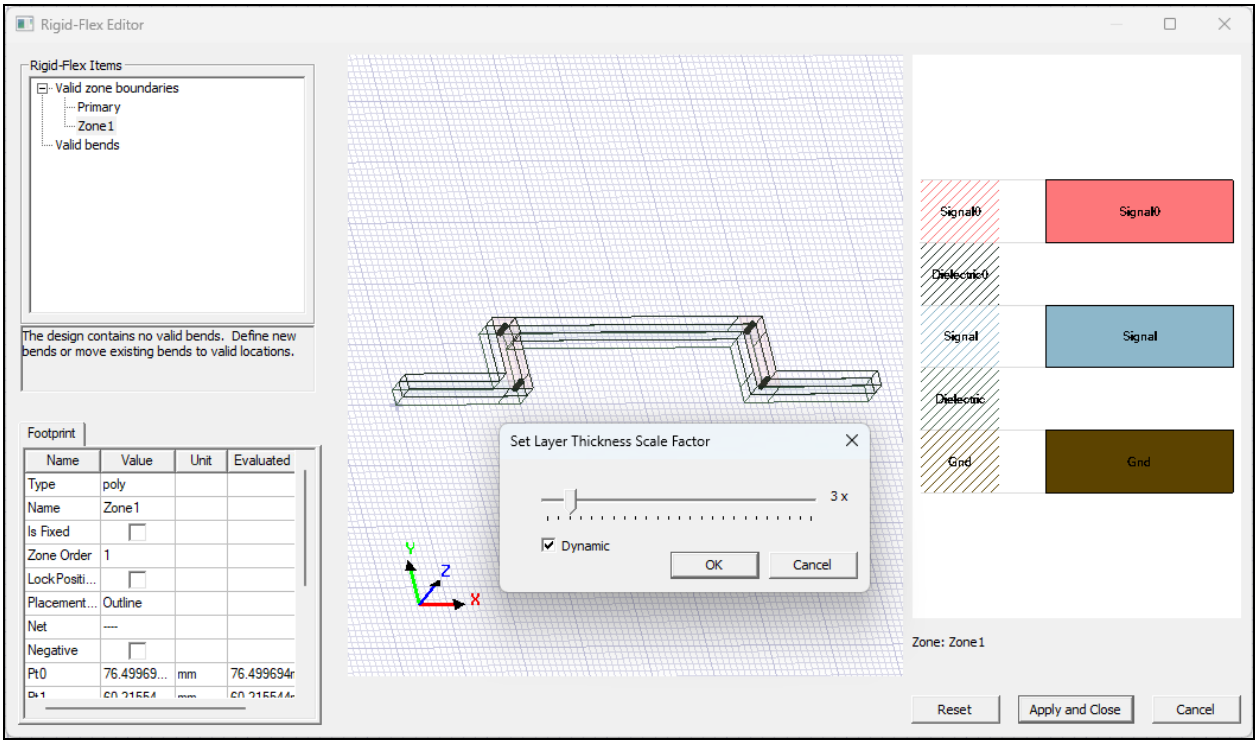
1. Right-click in the preview window and select **Stretch Z** to open the **Set Layer Thickness Scale Factor** window.

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



2. Use the slider to set the scaling factor to any discrete value between 1 and 25.

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



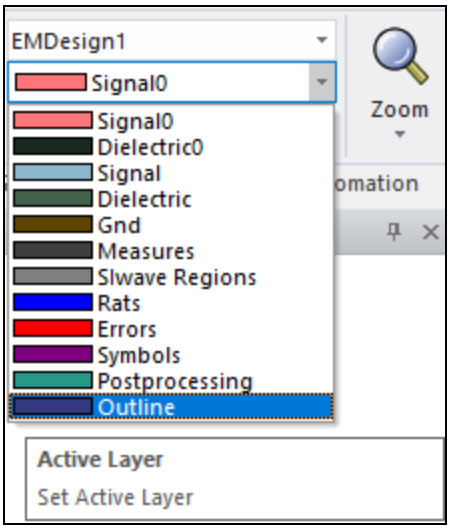
Continue to Defining Rigid-Flex Bends.

Defining Zones at the Layout Editor

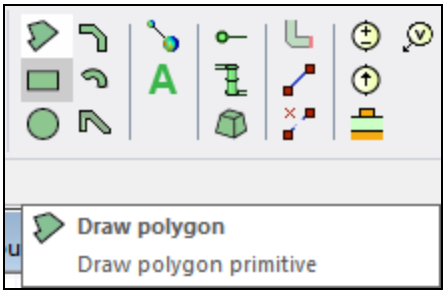
Follow these steps to outline and define zones from the **Layout Editor**.

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

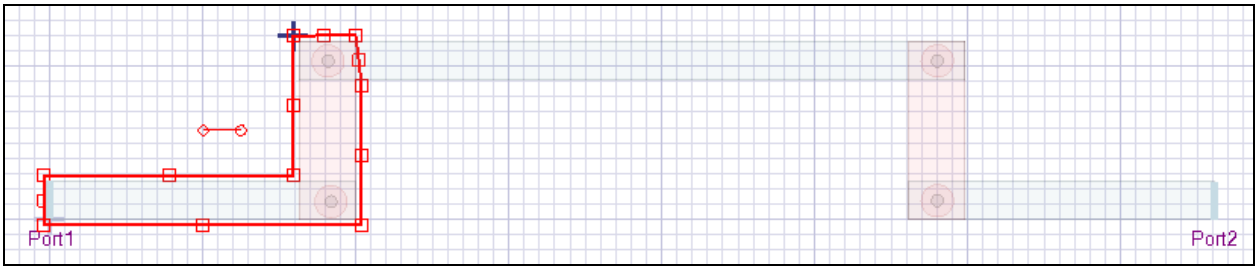
1. From the **Layout** tab, select **Outline** on the **Set Active Layers** drop-down menu.



2. From the **Layout** tab, select any primitive drawing tool (e.g., **Draw polygon**).



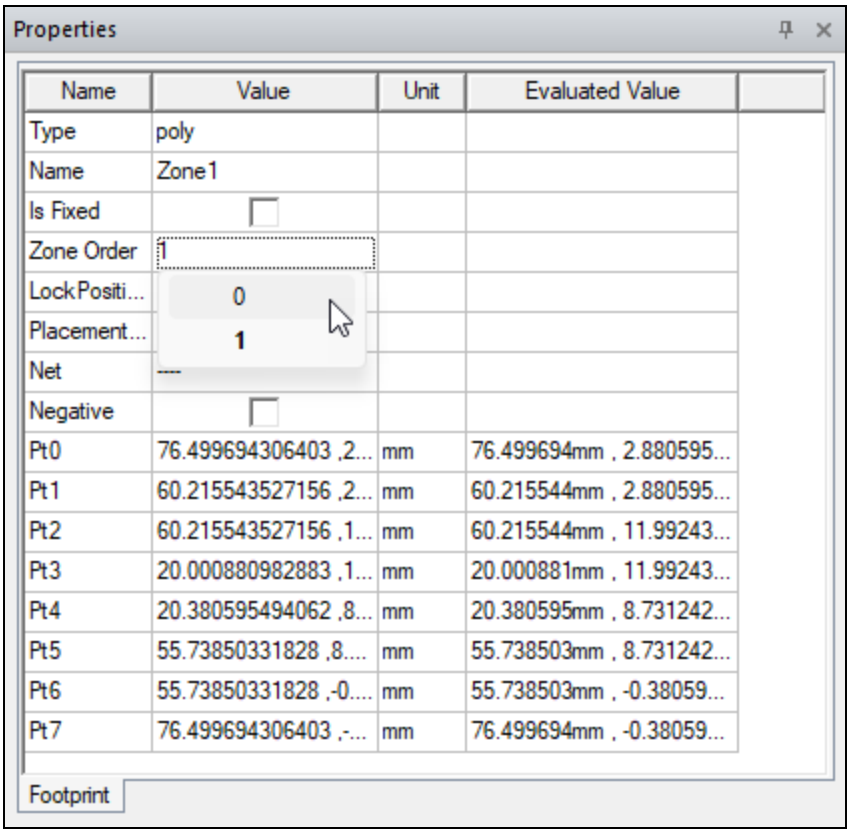
3. Draw a zone over the design using the same method described in the HFSS 3D Layout Getting Started Guides for drawing a primitive.



4. Repeat steps **2-3** for each zone. For best results, each successive zone should abut another zone.

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 a zone. From the **Properties** window, select the appropriate zone designation on the **Name Value** field's drop-down menu.



6. Once the primitive is assigned to a zone, the **Is Fixed** and **Zone Order** rows appear. If selected, do either or both of the following:
   - Check the box in the **Is Fixed Value** field to designate the primitive as fixed.
   - Assign the primitive a new **Zone Order** from the drop-down menu in the **Zone Order Value** field.
7. Repeat steps **5-6** for each zone.

Continue to Defining Rigid-Flex Bends.

Defining Rigid-Flex Bends

Follow these steps to create and define one or more rigid-flex bends using the **Rigid-Flex Workflow**.

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

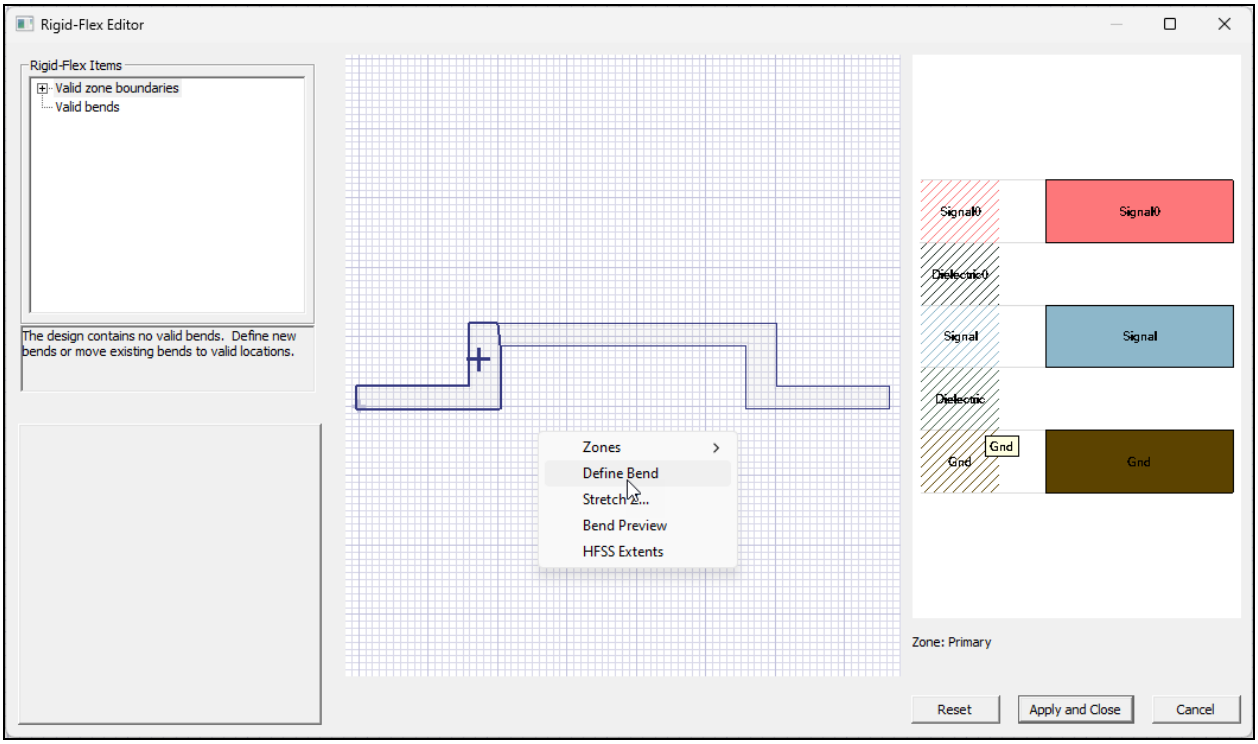
Note:

Although bends do not need to be placed within defined zones, there are other notable restrictions. Refer to "Bend Restrictions" on page 6-70.

Note:

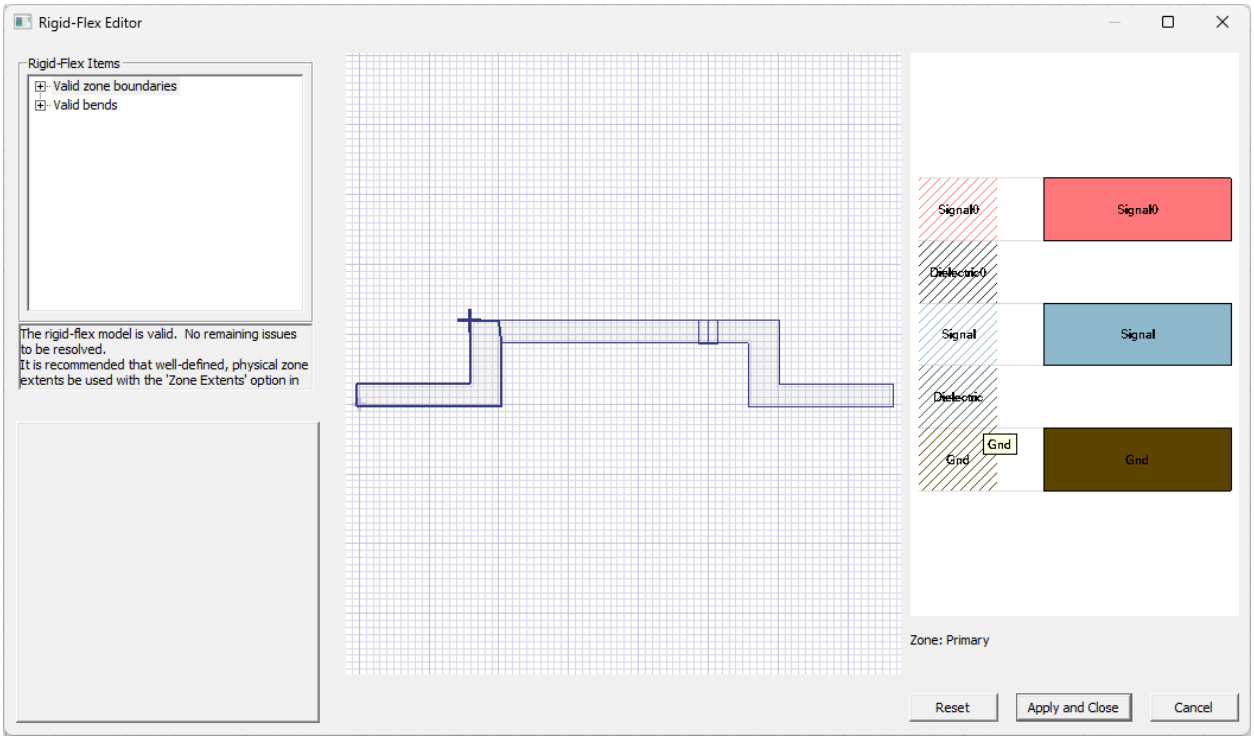
Bends can an also be defined at the **Layout Editor**. Refer to "Defining Bends at the Layout Editor" on page 6-74.

1. Right-click within the preview window and select **Define Bend**.



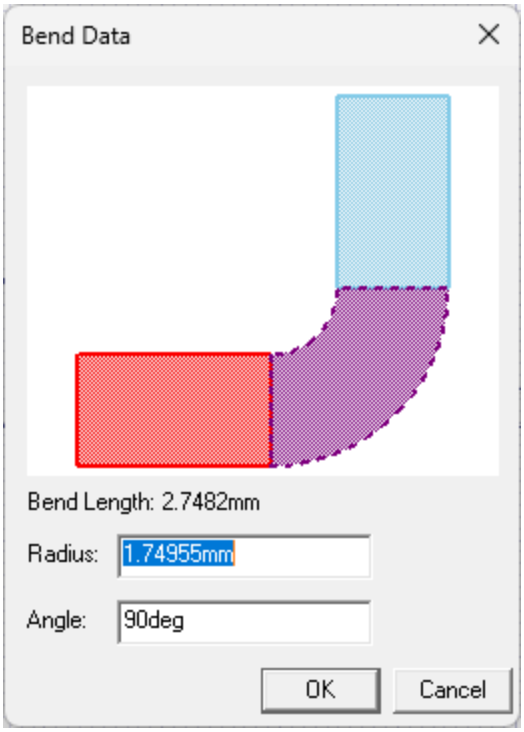
2. Click within any zone to begin placing a bend. Orient the bend, as appropriate, and click again to simultaneously place the bend and open the **Bend Data** window.

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



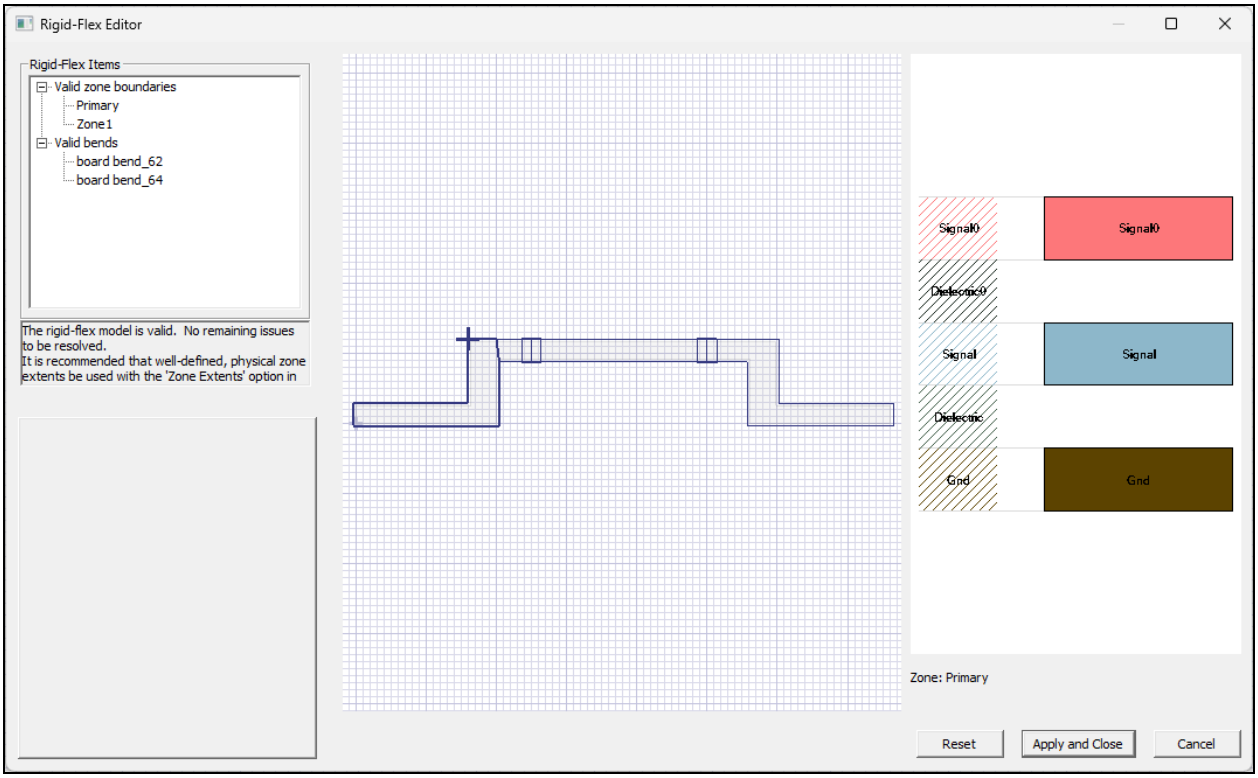
3. Enter the selected **Radius** (e.g., **3.181mm**) and **Angle** (e.g., **90deg**) in the appropriate fields.

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



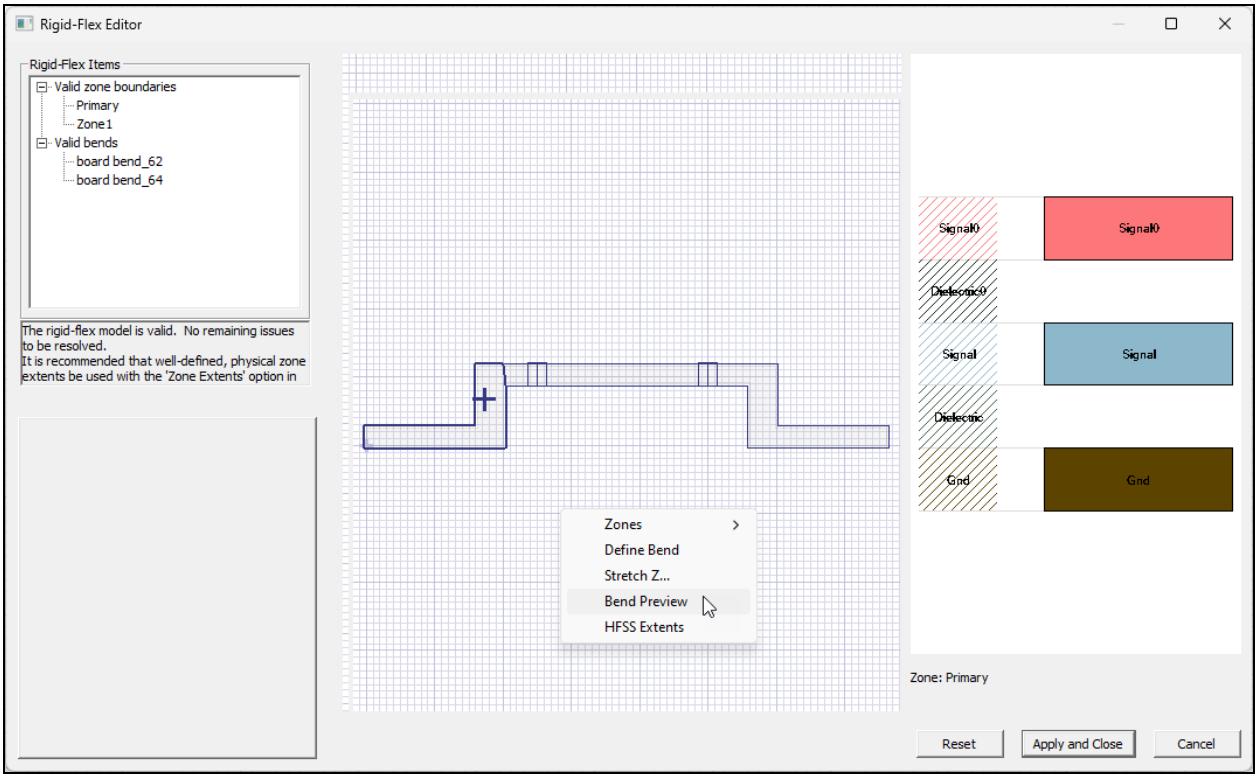
4. Click **OK** to close the **Bend Data** window and return to the **Rigid-Flex Editor** window.
5. Repeat steps **1-4** to create more bends, as appropriate.

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



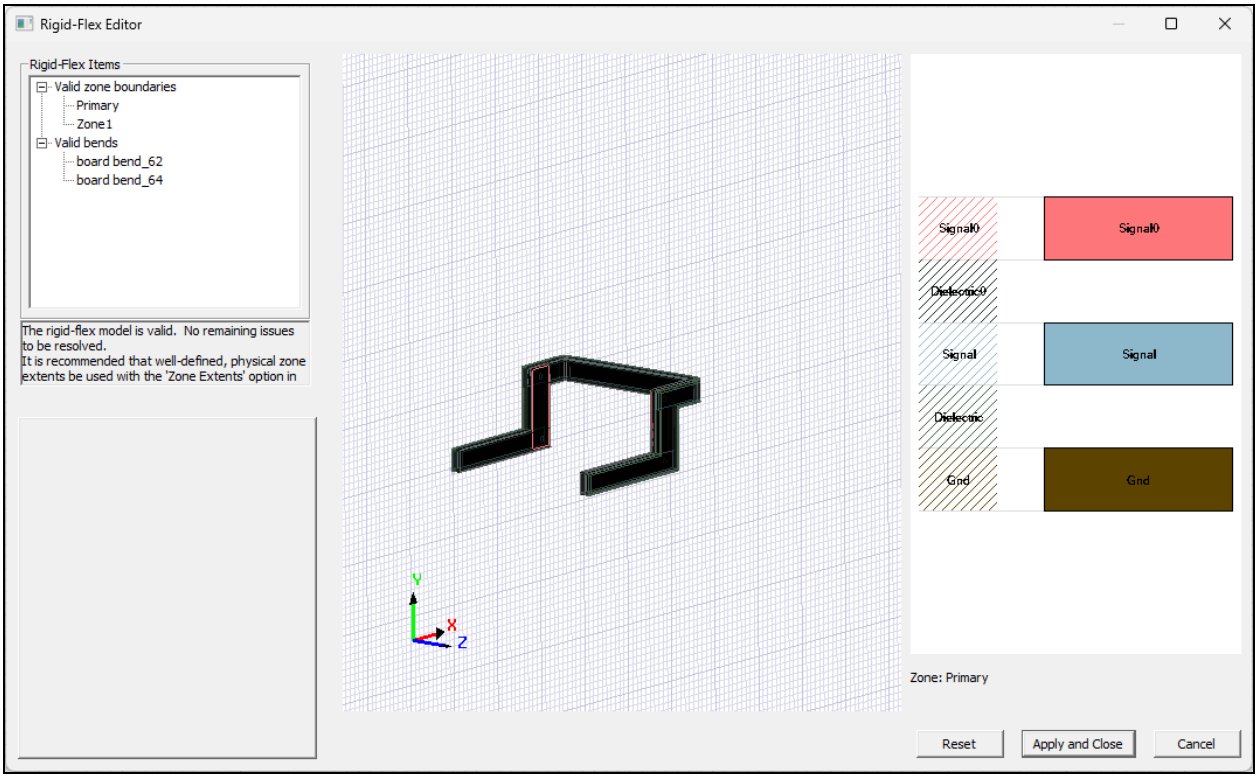
6. Right-click within the preview window and select **Bend Preview** to see the rigid-flex version of the design model.

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



7. Reorient the model using the same control method as in the **Layout Editor**. Refer to the HFSS 3D Layout Getting Started Guides, if applicable.

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

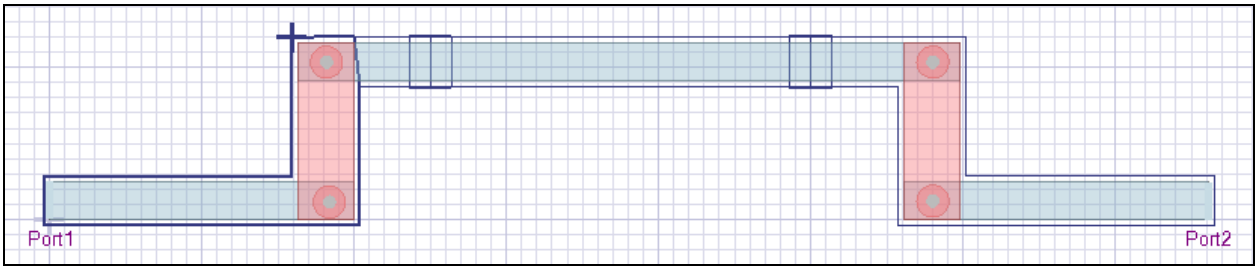


Note:

If bends appear incorrect in the UI (e.g., missing geometry in the bend, the bend (s) not visually applied whatsoever, et cetera), try changing the location of the fixed region. Certain UI-related issues are purely graphical and do not affect meshing or simulation results. The bend visualization is a lightweight representation of the deformed geometry. To verify the actual geometry used in simulation, plot the mesh or display pre-process geometry. If the design is in a bent state, the mesher is selected automatically, regardless of user-selection. Refer to Meshing Method.

8. Click **Apply and Close** to close the **Rigid-Flex Editor** window. The new zones and bends appear in the **Layout Editor**.

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

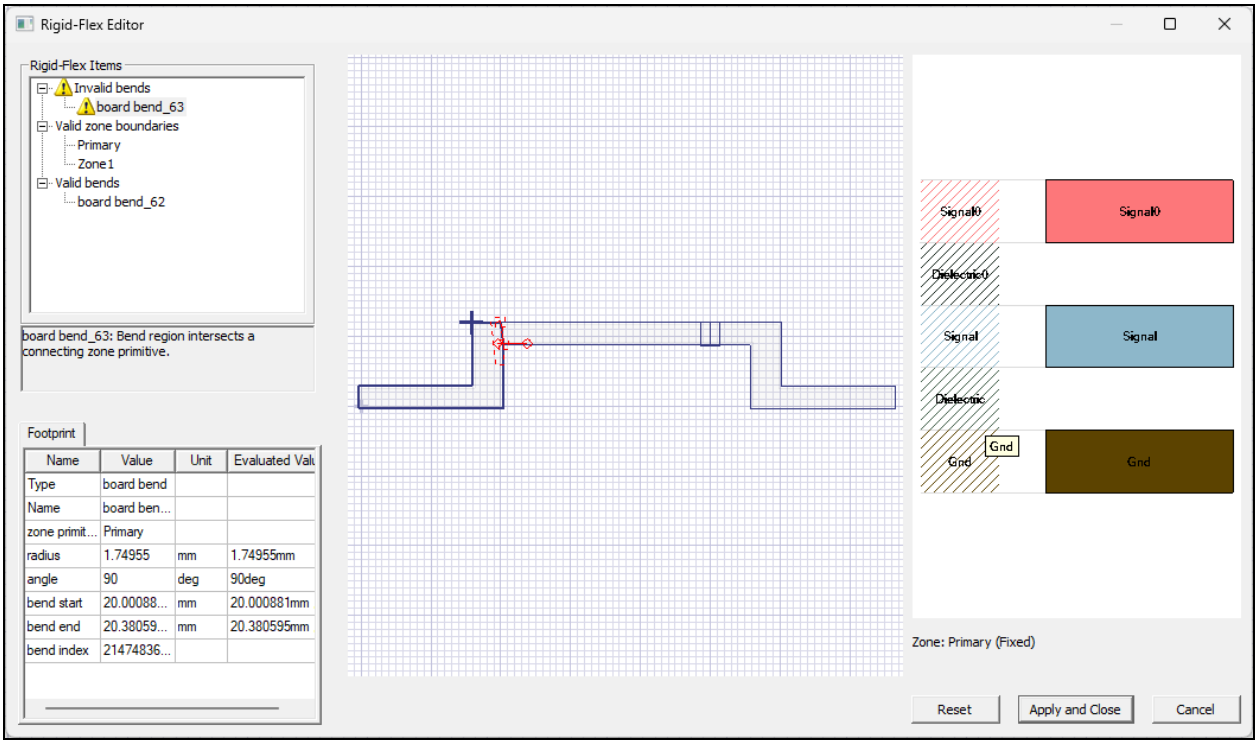


Continue to Analyzing a Design With Rigid-Flex Bends.

Bend Restrictions

Bends cannot overlap the anchor point, one another, ports or other components, or be placed across multiple zones. Bends are defined by two points in the x-y plane indicating their placement in the model, a radius value indicating the sharpness of the bend, and an angle. The magnitude of the angle indicates the extent of the wrapping while the sign of the angle indicates the bend direction (i.e., positive or negative angle implies a positive-z or negative-z bend direction respectively). When computing bent geometry, the software defines the neutral root as a point on the face where the bent region connects with the unbent region. To make the deformation consistent between positive and negative bends, the elevation of the neutral root is dictated by the sign. If the bend is positive, the neutral root is placed at the bottom of the zone stackup. If the bend is negative, the neutral root is placed at the top of the zone stackup. For geometry in bend regions at the same elevation as the neutral root, in-plane distances are preserved, but geometry is contracted at other elevations. The minimum bend radius is 1.1x the thickness of the zone on which the bend is placed. Invalid placement of bends results in error messages, which appear in the **Rigid-Flex Items** area.

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 bend with a bend radius that is too small can severely deform the model. The bends shown in the software are geometric deformations only; no mechanical physics are simulated. When a bend is applied, the outer surface of the bend becomes the reference and remains undistorted, while the inner surface is distorted to accommodate the curvature.

For example, consider a stack-up with a total thickness of 0.12mm.

Name	Type	Material	Thickness	Primary	Zone1
Signal	signal	copper	0.01mm	☑	☑
Dielectric	dielectric	FR4_epoxy	0.1mm	☑	☑
Gnd	signal	copper	0.01mm	☑	☑

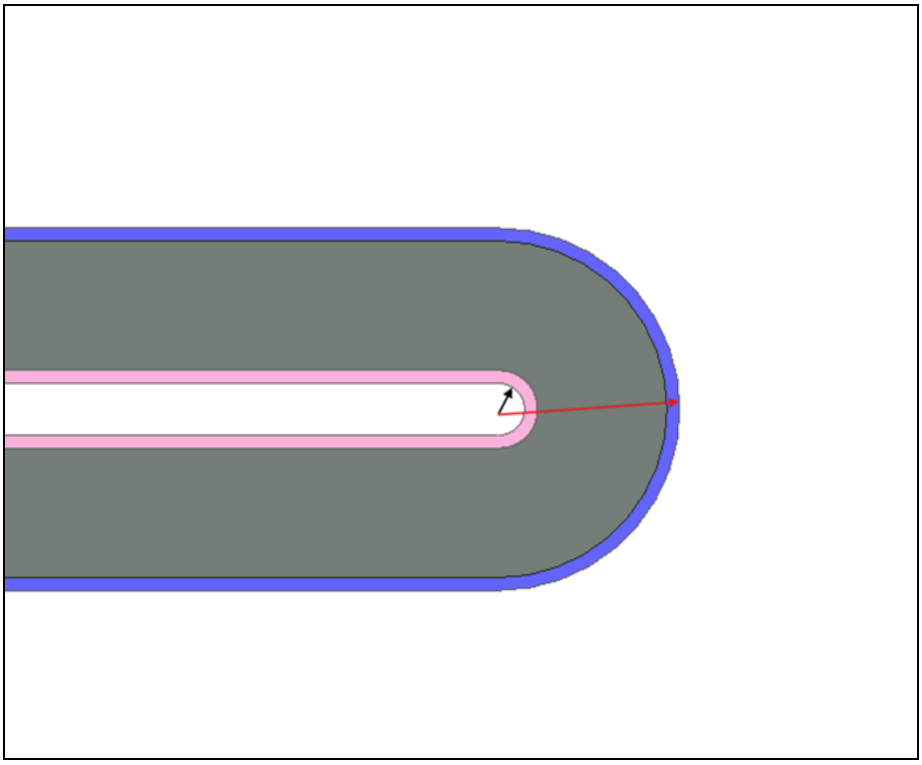
A 180° bend with a radius of 0.14mm is applied to the stackup (red arrow).

The length of the outer surface of the bend will be: $\pi \times 0.14\ \text{mm} \approx 0.4398\ \text{mm}$

This outer surface remains undistorted. However, the inner radius of the bend becomes: 0.14 mm – 0.12 mm = 0.02 mm (black arrow)

This means the inner (top) signal layer must compress to a length of: $\pi \times 0.02\ \text{mm} \approx 0.0628\ \text{mm}$

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



This results in more than a 7× contraction relative to the flat state, which can produce highly distorted geometry and lead to simulation issues.

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

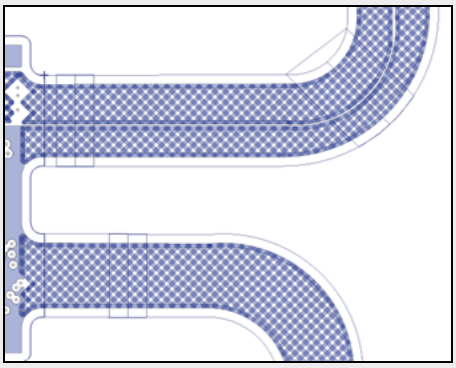
Note:

Setting a Defeaturing Tolerance (HFSS > Advanced Setup Only)

If arcs are defined with dense faceting in a design, consider applying a defeaturing tolerance to simplify the arc geometry. By enable **Use polygon defeaturing** and setting a **Tolerance as a ratio of the data extent** (available from the HFSS setup window on the **Advanced** tab), very close points, collinear points, and thin intrusions that do not contribute meaningfully to the geometry are removed during simulation. This is particularly useful if a design has highly faceted curves or geometry with excess detail. This can significantly reduce computational complexity and improve solver performance, especially during mesh generation.

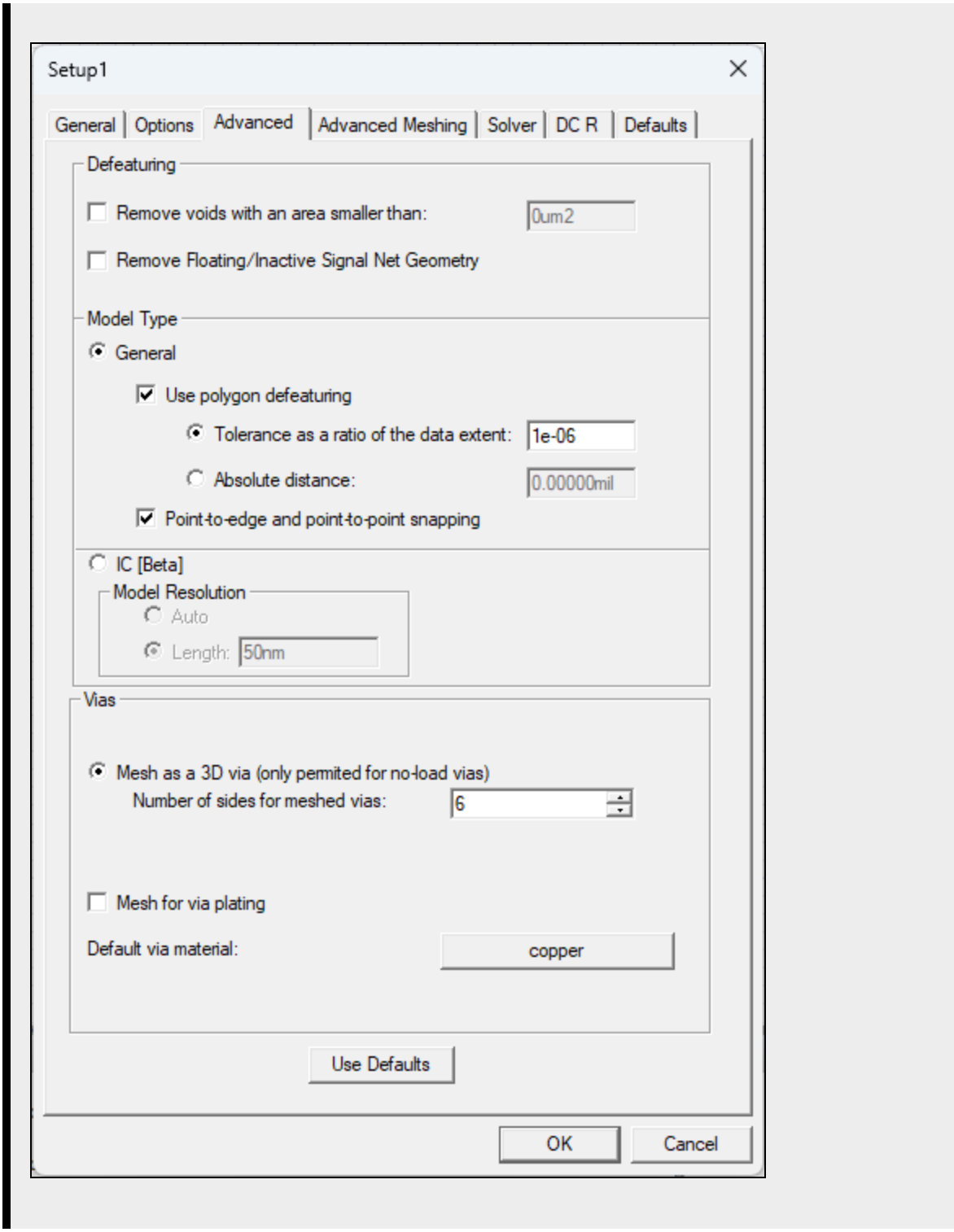
Removing Voids (HFSS > Advanced Setup Only)

Another option is to automatically defeature or remove voids contained inside bends, particularly when there is hatched geometry in the design. From the **Advanced** tab, check the **Remove voids with an area smaller than** box and then specify a value. Users should aim for balanced void removal, as excessive internal detail can contribute to mesh failures. While not all voids need to be eliminated, applying a sensible void removal tolerance can improve meshing robustness and overall simulation success.



To set a defeaturing tolerance or remove voids smaller than a specified value, refer to the Advanced tab in the HFSS setup window.

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

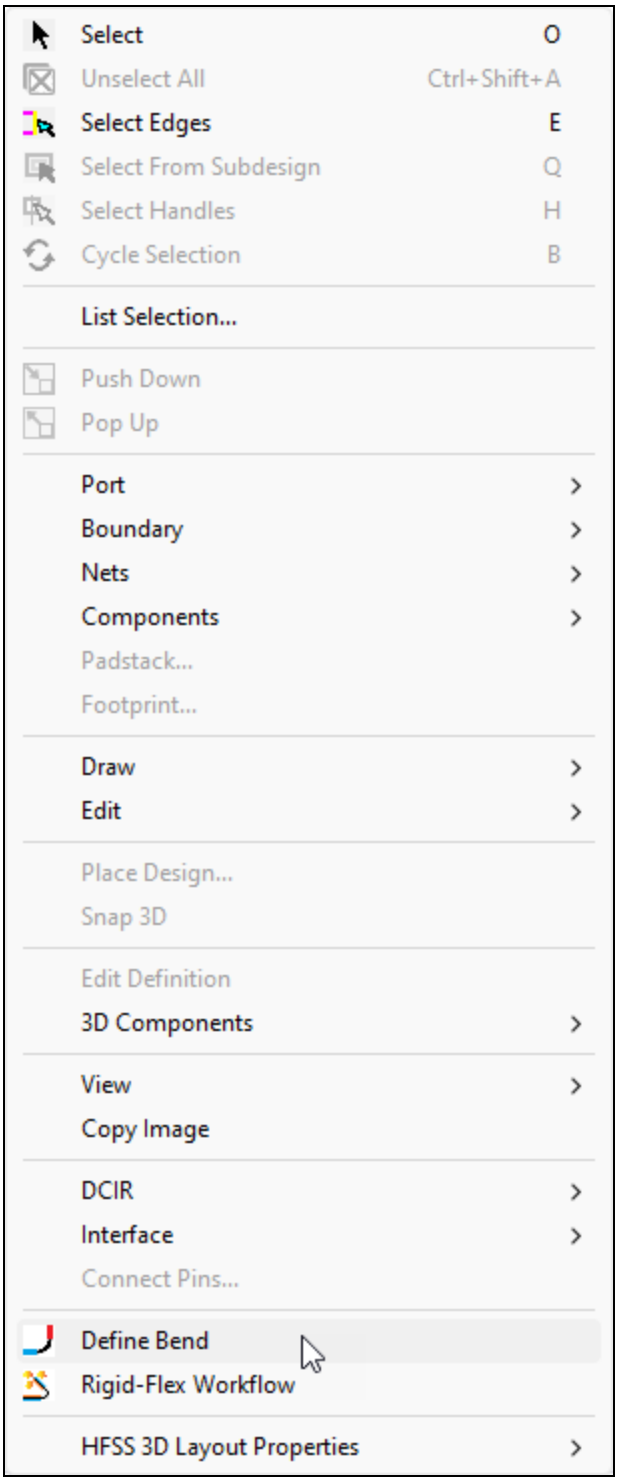


Defining Bends at the Layout Editor

Follow these instructions to create and define bends within the **Layout Editor**.

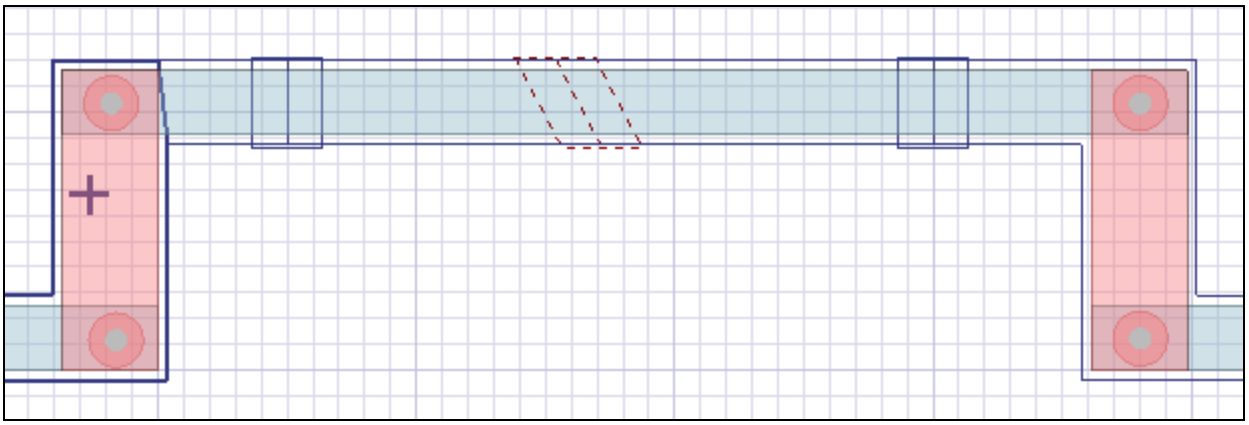
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From within the **Layout Editor**, right-click anywhere and select **Define Bend**.

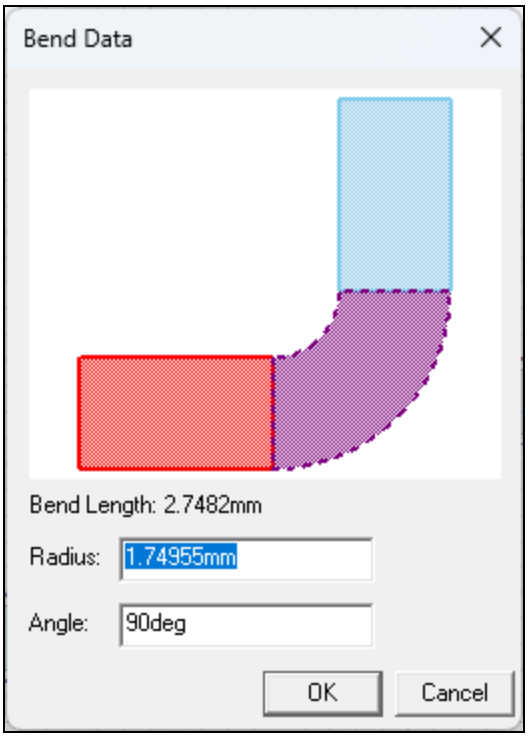


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click within any zone to begin placing a bend. Orient the bend, as appropriate, and click again to simultaneously place the bend and open the **Bend Data** window.



3. Enter the selected **Radius** (e.g., **1.74955mm**) and **Angle** (e.g., **90deg**) in the appropriate fields.



4. Repeat steps **1-3** to create more bends, as appropriate.

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

Note:

Rigid-Flex Bends can be animated. Refer to Animating Rigid-Flex Bends.

Continue to Analyzing a Design With Rigid-Flex Bends.

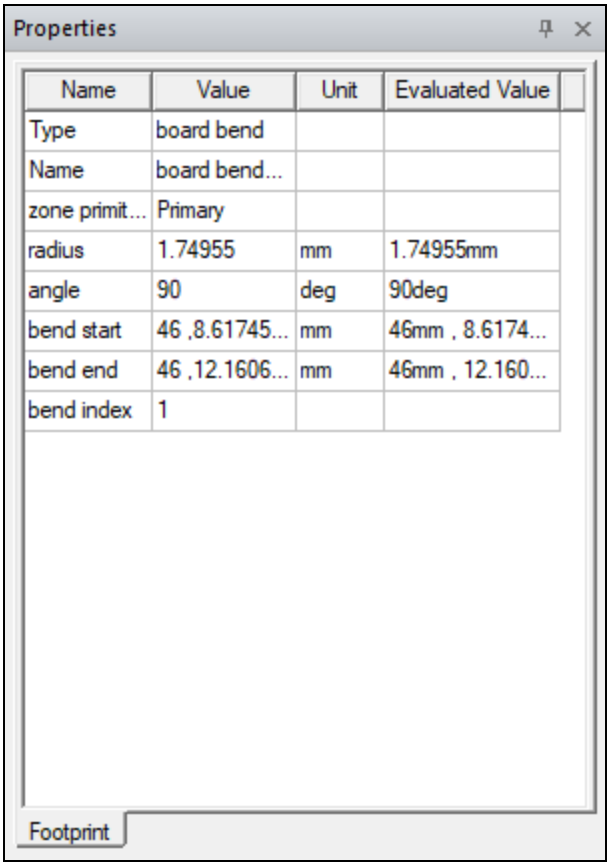
Animating Rigid-Flex Bends

Follow these steps to animate rigid-flex bends.

Note:

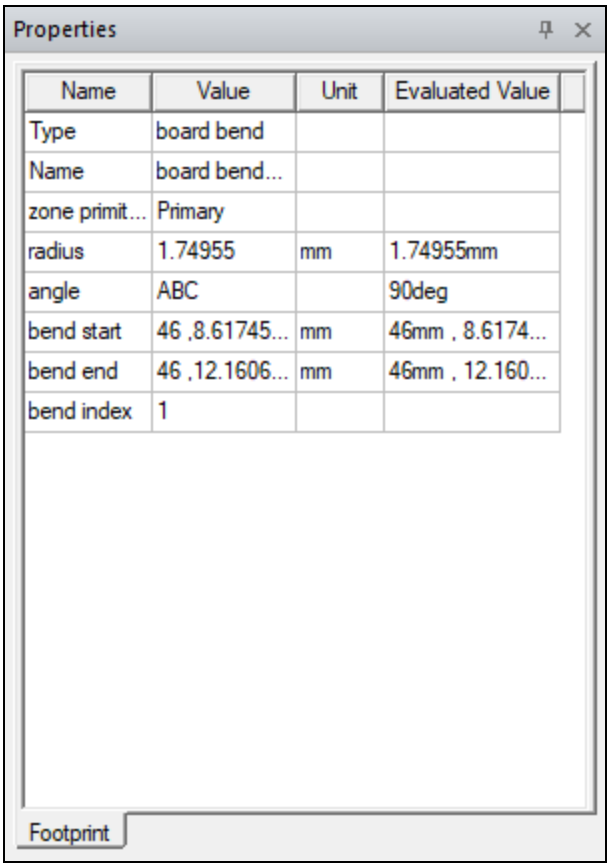
Before continuing, ensure rigid-flex are defined. Refer to "Defining Bends at the Layout Editor" on page 6-74.

1. From the **Layout Editor**, select a rigid-flex bend to populate the **Properties** window with its parameters.



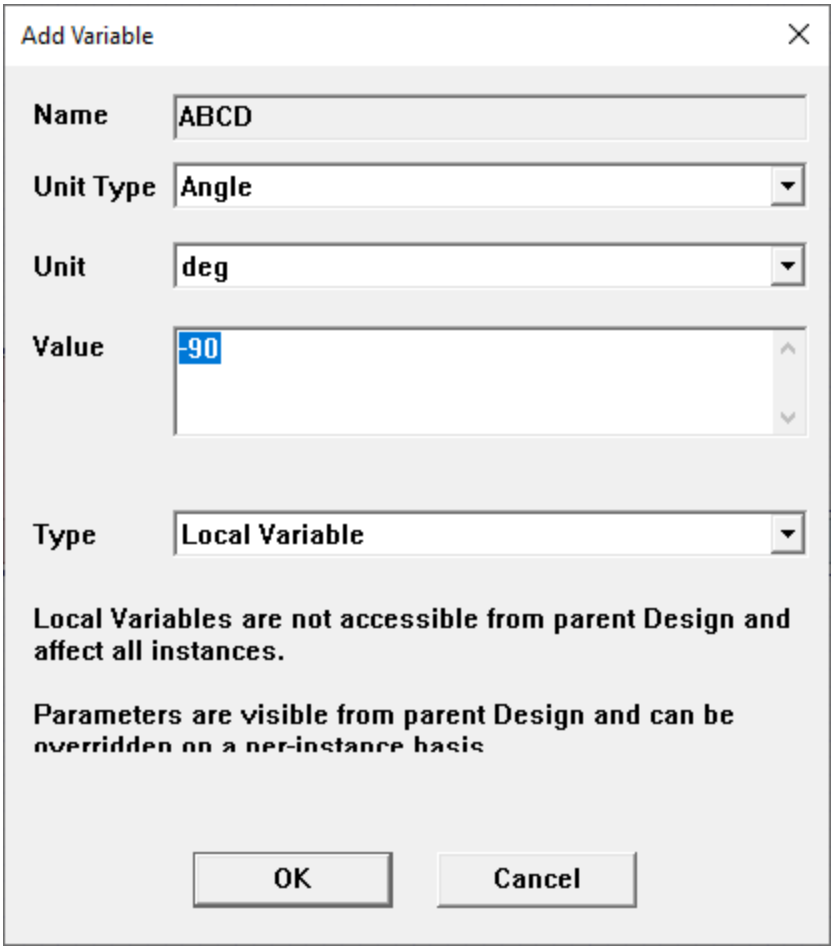
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter a variable in the angle field (e.g., **ABC**) and press **Enter** to open the **Add Variable** window.

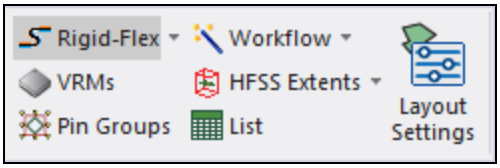


3. Enter an integer (positive or negative) in the **Value** field (e.g., **-90**) and **Angle**.

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

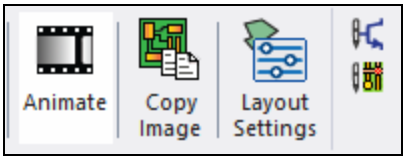


4. Click **OK** to close the **Add Variable** window and return to the **Rigid-Flex Editor** window.
5. Repeat steps **1-4** for each bend.
6. From the **Layout** tab, click **Rigid-Flex** to apply the rigid-flex bend to the design.

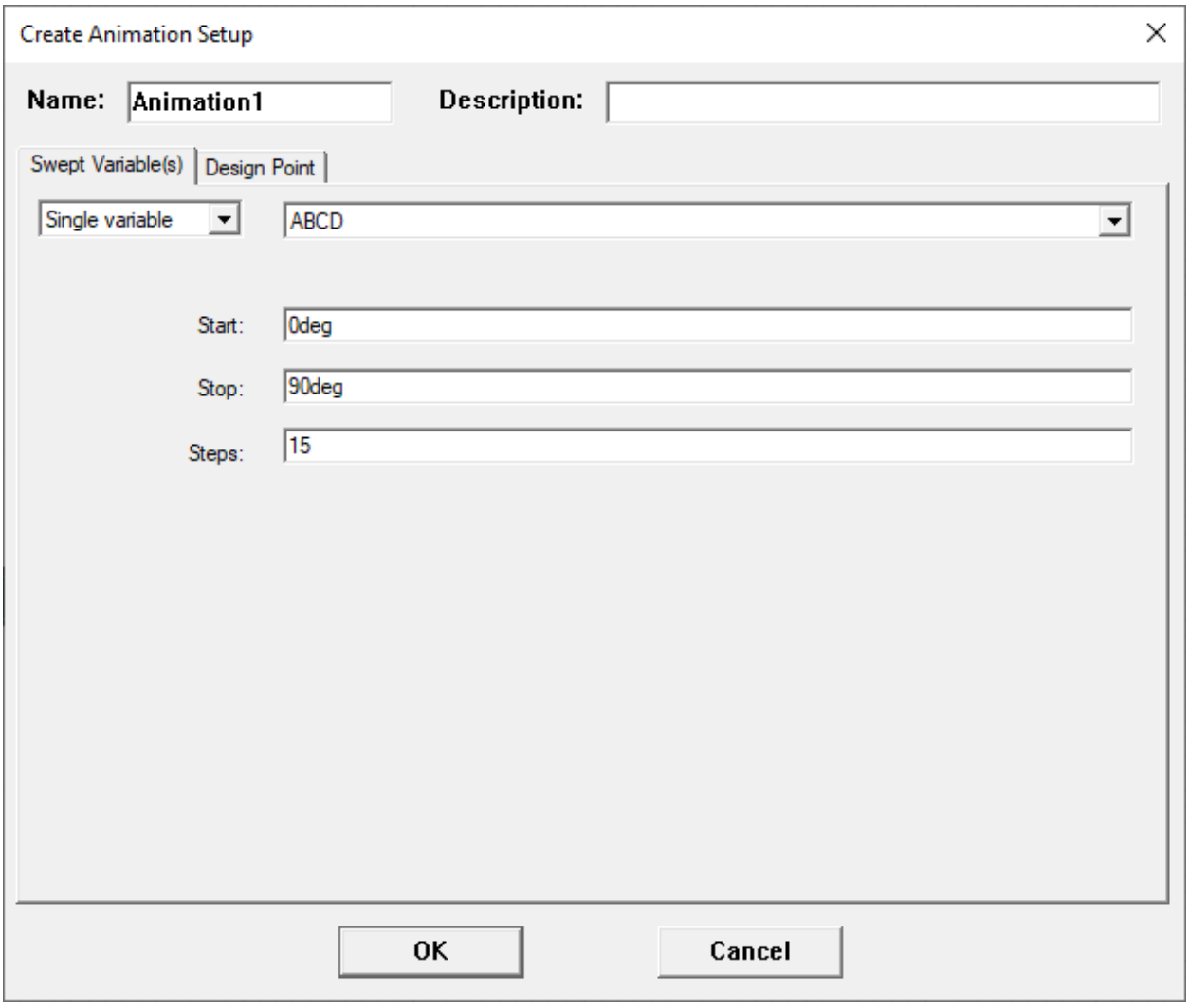


7. Navigate to the **View** tab and select **Animate** to open the **Create Animation Setup** window.

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

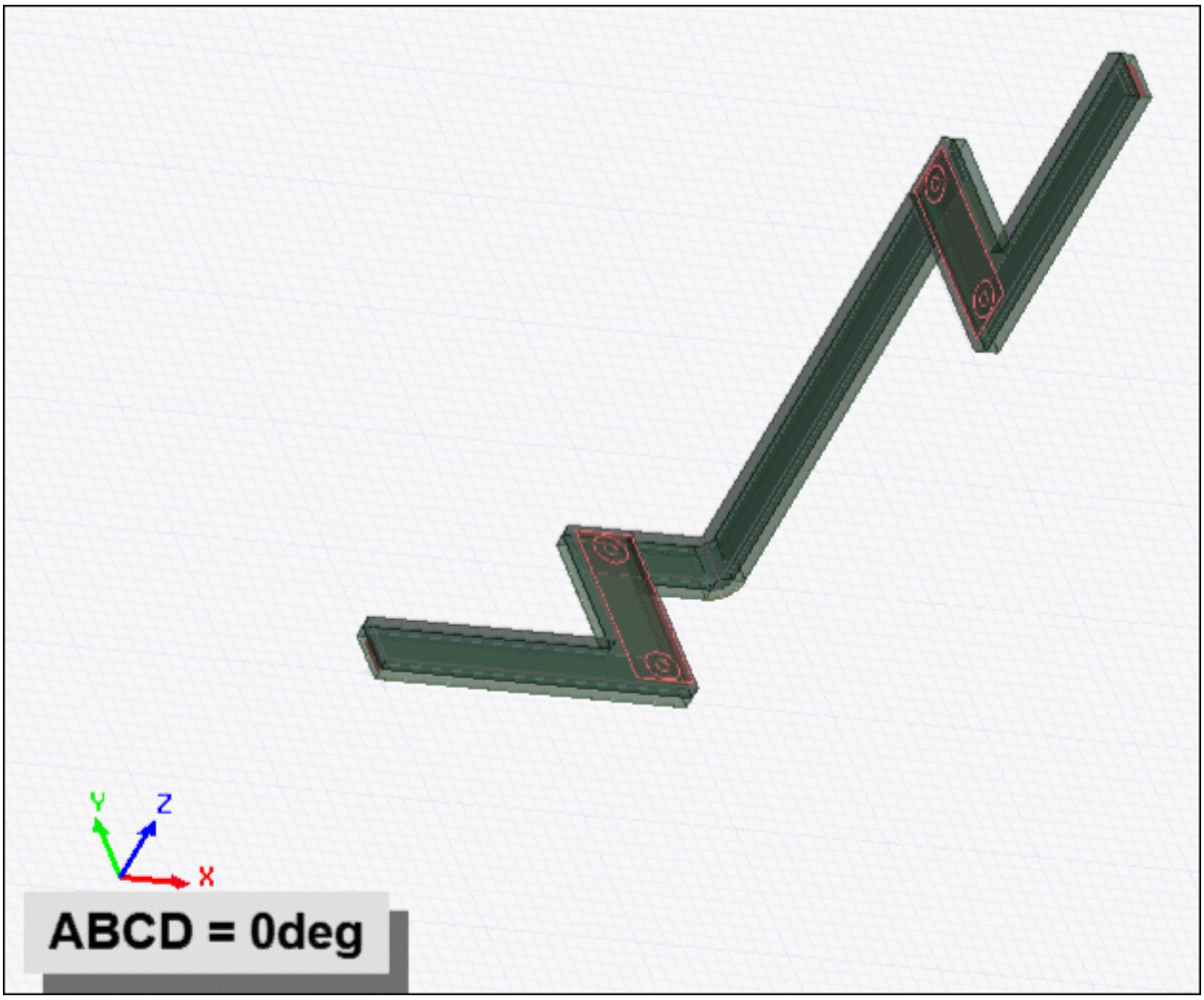


8. From the **Create Animation Setup** window, do the following:
    - Select the new variable (e.g., **ABC**) from the **Single variable** drop-down menu.
    - Enter an appropriate **Start** angle (e.g., **0deg**).
    - Enter an appropriate **Stop** angle (e.g., **90deg**).



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 create the animation. Refer to Creating Animations.



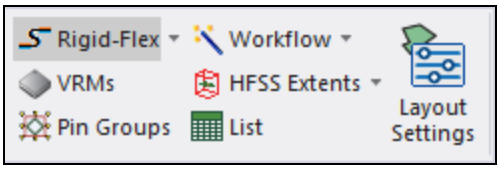
Continue to Analyzing a Design With Rigid-Flex Bends.

Analyzing a Design With Rigid-Flex Bends

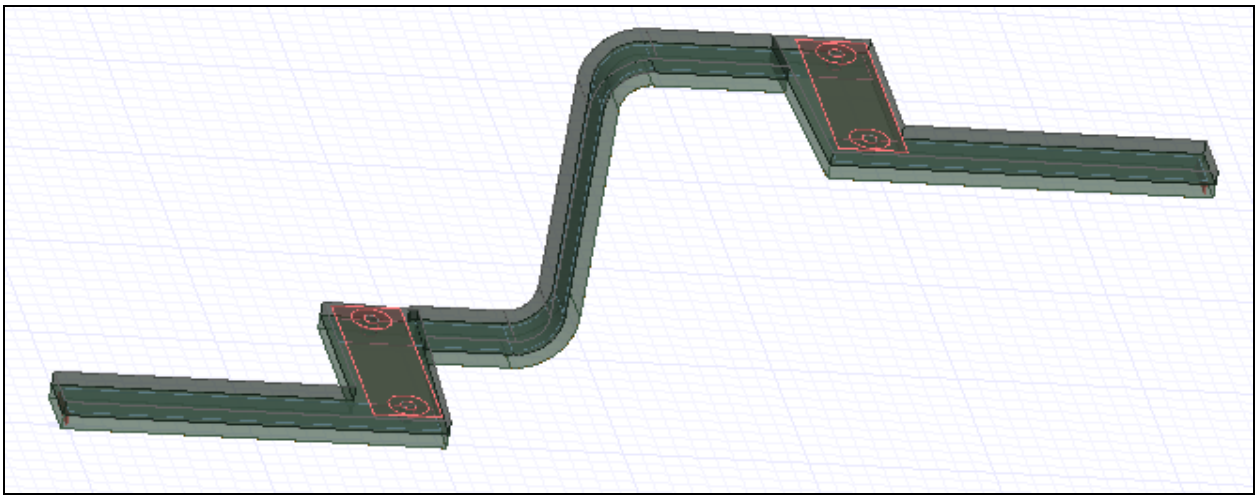
Once rigid-flex bends have been defined, follow these steps to analyze the design in both its bent and flat states.

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 **Layout** tab, click **Rigid-Flex** to apply the rigid-flex bend to the design.



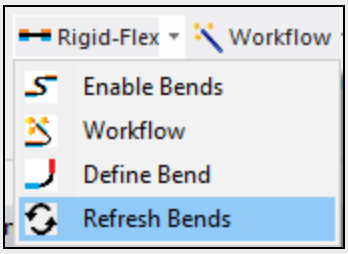
   The design in the **Layout Editor** will change to the rigid-flex version. Reorient, as appropriate.



2. Now that the bent model is generated, click **Rigid-Flex** to toggle between the bent and flat designs.

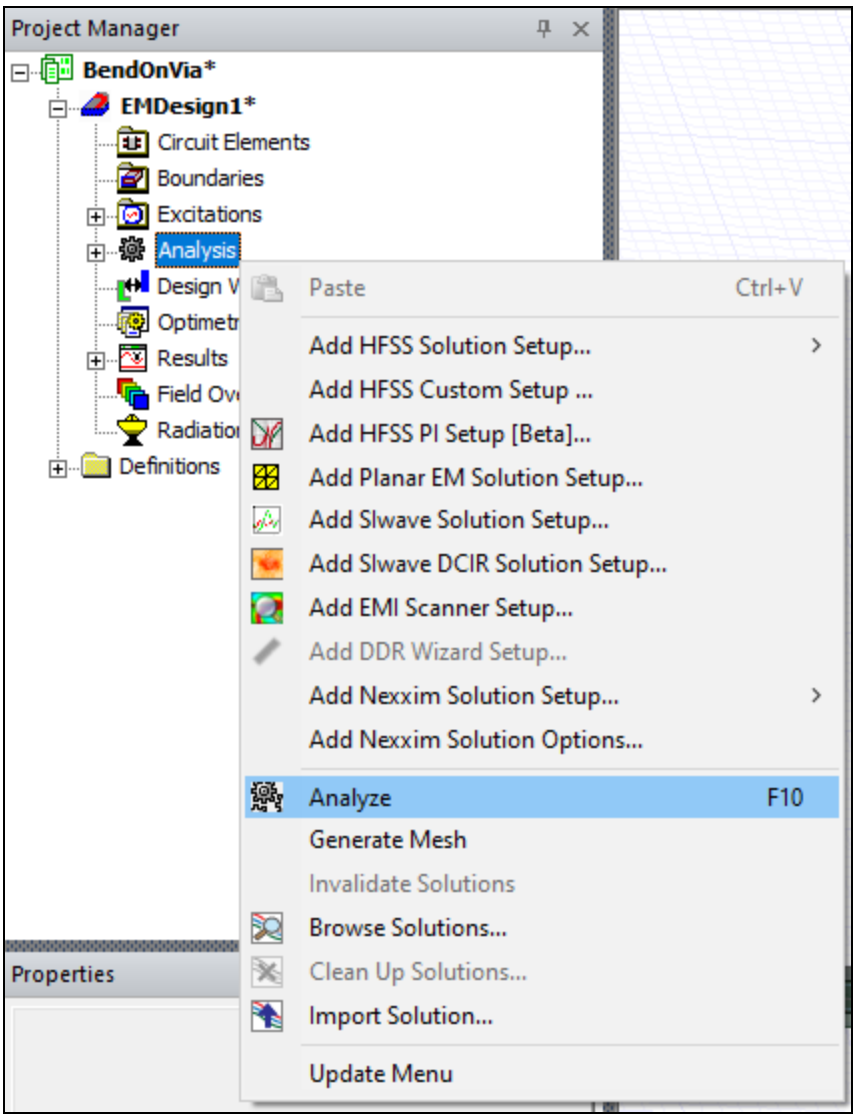
Note:

To return to the **Rigid-Flex Editor** window, click the **Rigid-Flex** arrow and select **Workflow**. If the design is altered, select **Refresh Bends** to overwrite the saved rigid-flex design.



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

3. **Set up a Solution** to analyze. Refer to the Solution Setup in HFSS 3D Layout.
4. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** and select **Analyze**. Wait for the analysis to complete.



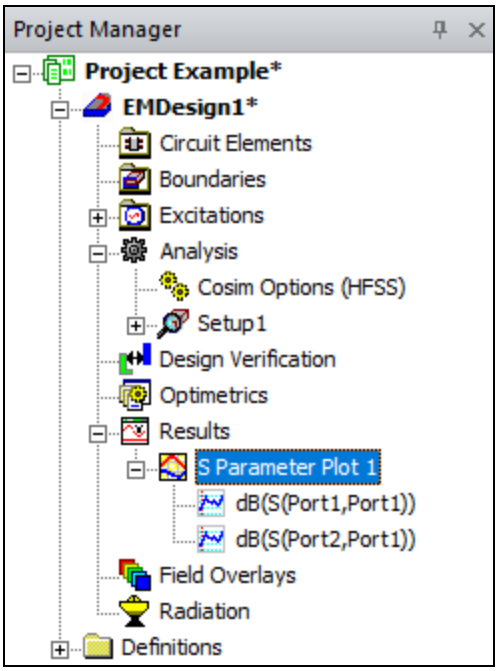
5. Click **Rigid-Flex** to change the model from the bent version to the flat version or vice versa. Then, on the **Project Manager** window, right-click **Analysis** and select **Analyze**, again.

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 second time the design is analyzed is no more expedient because the entire analysis is conducted again for the design in its changed state. The solutions derived on the analyses do not invalidate one another.

6. From the **Project Manager** window, expand the **Project Tree** > **[*Active Design Folder*]** > **Results** > plot folder (e.g., **S Parameter Plot**). There is a solution for each (bent and flat) state.



7. Manually alter the model or bend angle to analyze different design variations, or parameterize the model to analyze and animate the design (refer to Design Variables and Animating Rigid-Flex Bends).

Deactivating Simplified Bend Graphics

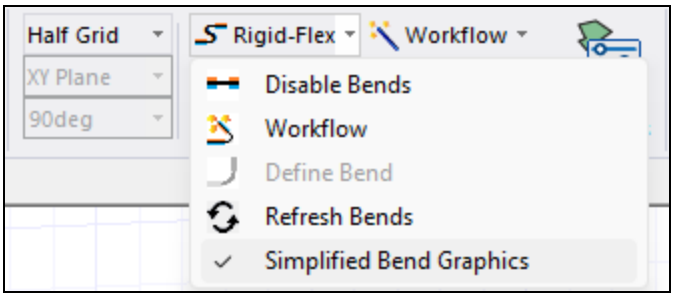
Depending on the complexity of the design, rendering a rigid-flex bend in the **Layout Editor** can take time and system resources. By default, **Simplified Bend Graphics** is active to more efficiently render the design. The basic shape of the design is maintained, but detail is not rendered. Follow these steps to deactivate **Simplified Bend Graphics**.

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

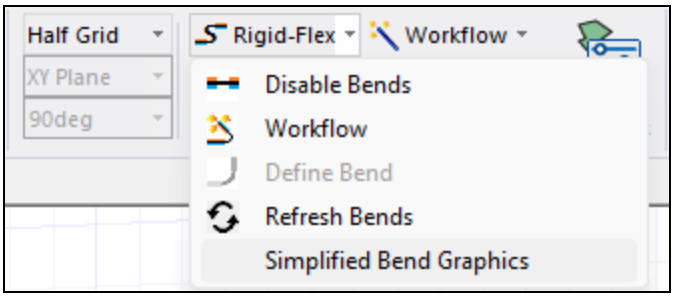
Note:

Simplification only affects rendering within the UI. It does not impact export of the bent design or simulation.

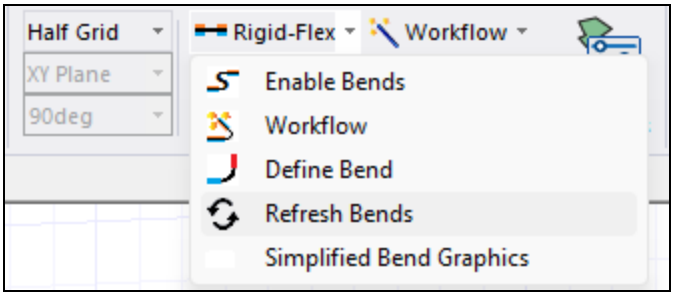
1. From the **Layout** tab ribbon, navigate to **Rigid-Flex** drop-down menu and select **Simplified Bend Graphics**.



The **Simplified Bend Graphics** check disappears.

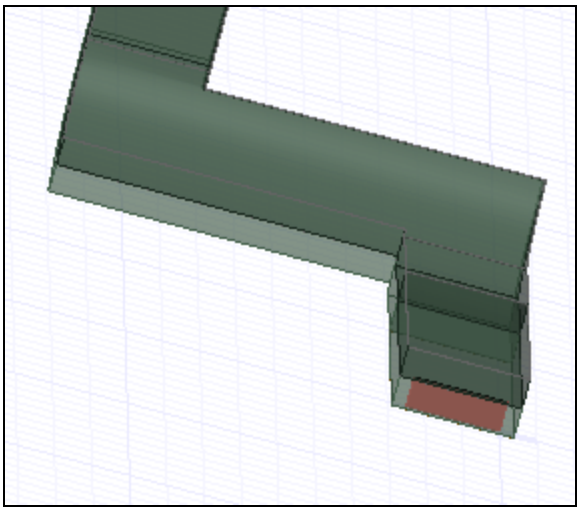


2. From the **Rigid-Flex** drop-down menu, select **Refresh Bends** to see the full rendering of the design in the **Layout Editor**.

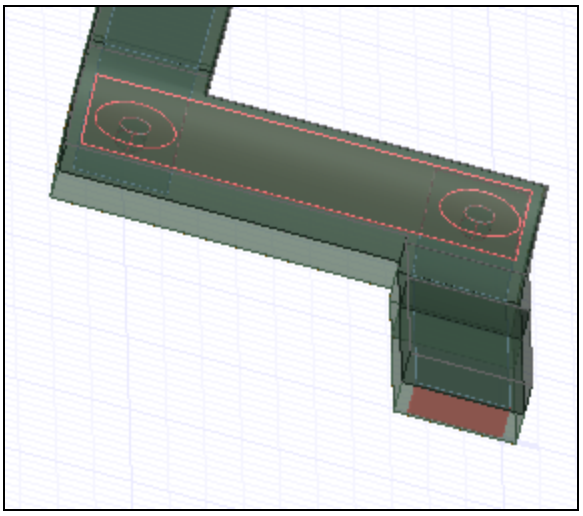


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Before **Simplified Bend Graphics** is deactivated, the model may look similar to the following example, maintaining the basic shape of the design.



After **Simplified Bend Graphics** is deactivated, the model will render completely.



3. To reactivate **Simplified Bend Graphics**, simply repeat steps **1**-**2**.

Continue to Exporting a Flexed Design Component

Exporting/Importing a Flexed Design Component

Follow the steps in these sections to export and import a design element with a rigid-flex bend as a component.

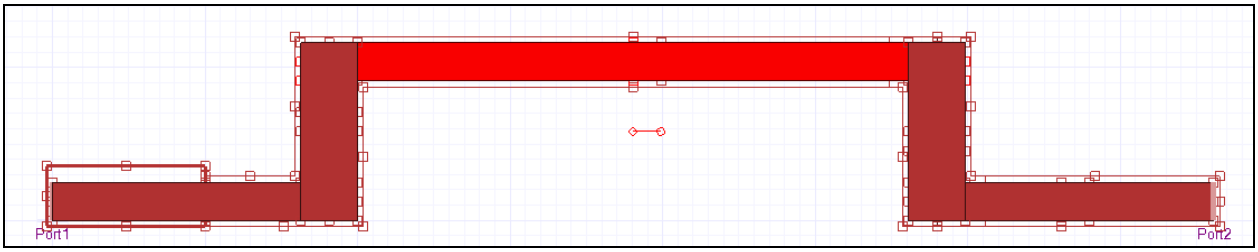
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Components with defined rigid-flex bends can be imported into a project as long the project does not already contain rigid-flex data.

Exporting a Flexed Design Component

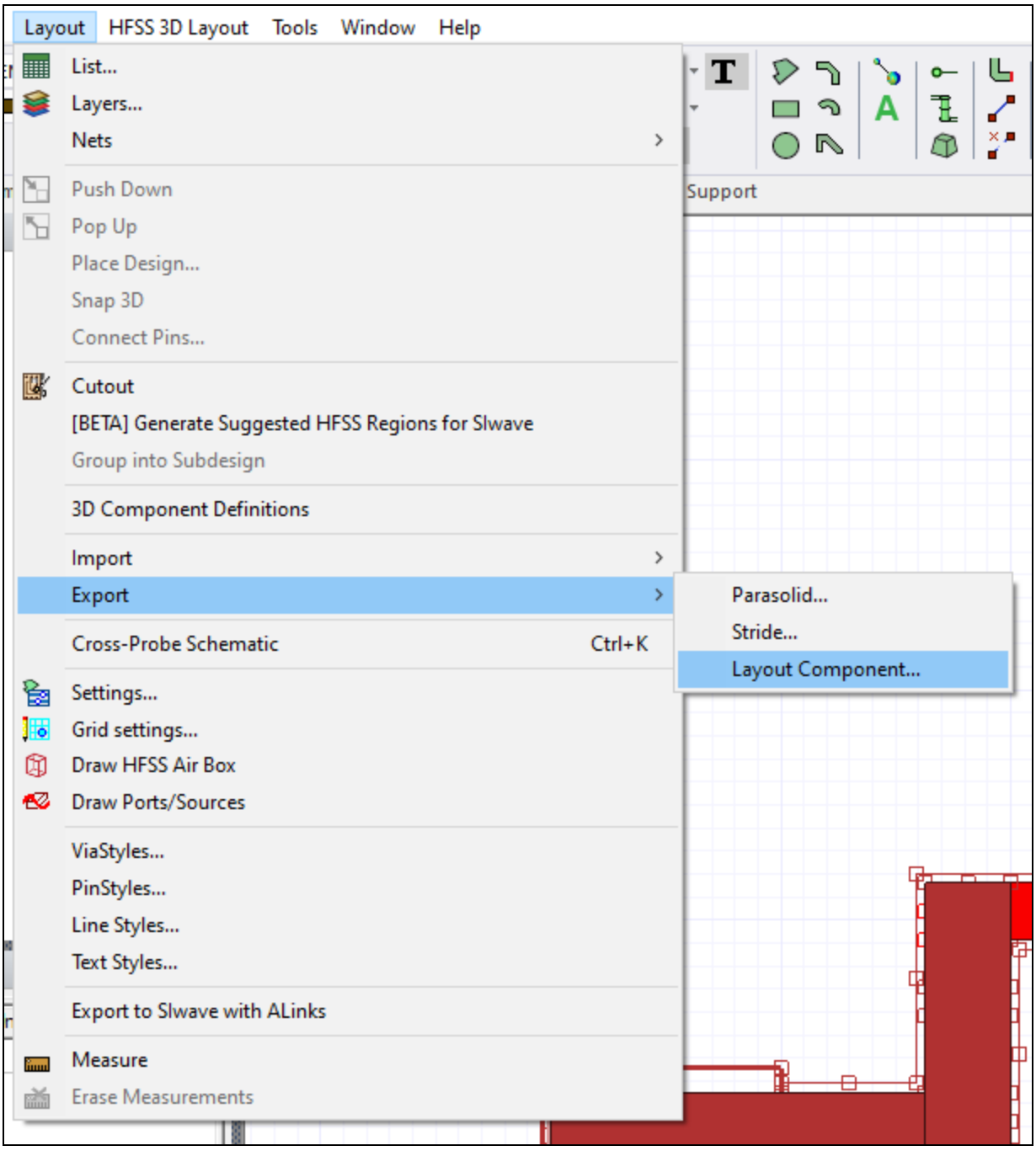
1. Select the design element(s) to export.



2. Navigate to **Layout** and select **Export** > **Layout Component** to open an explorer

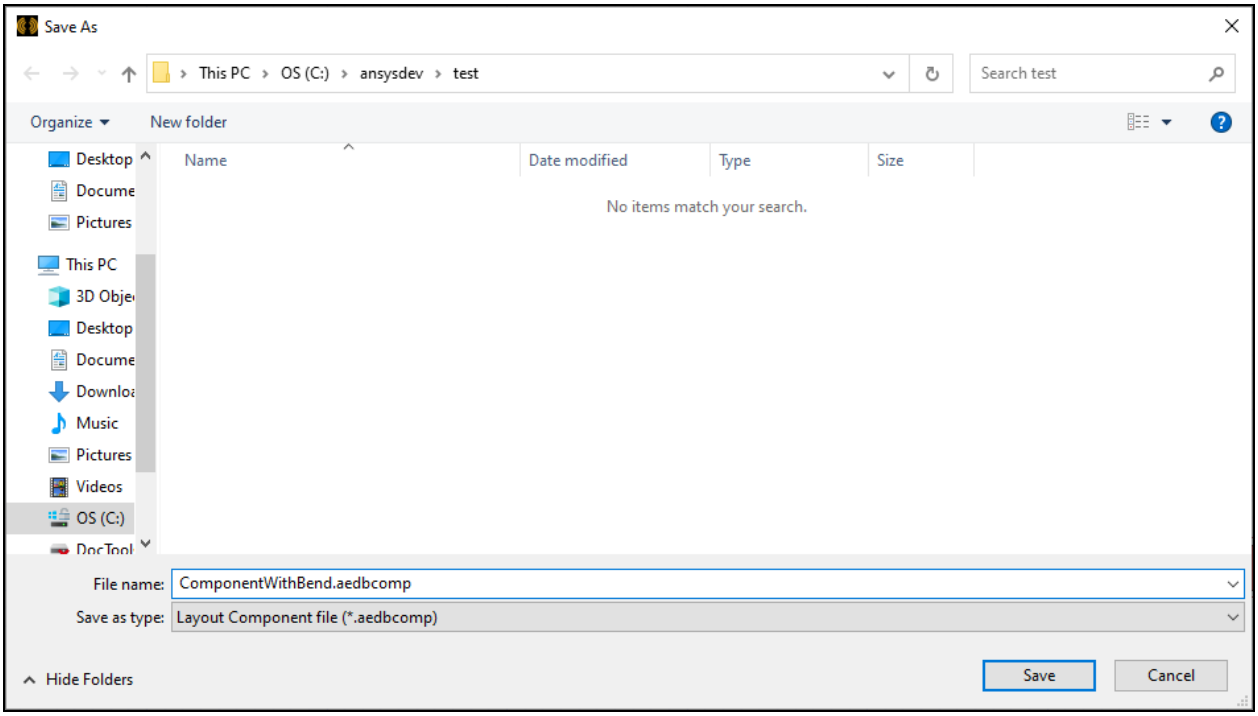
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

window.



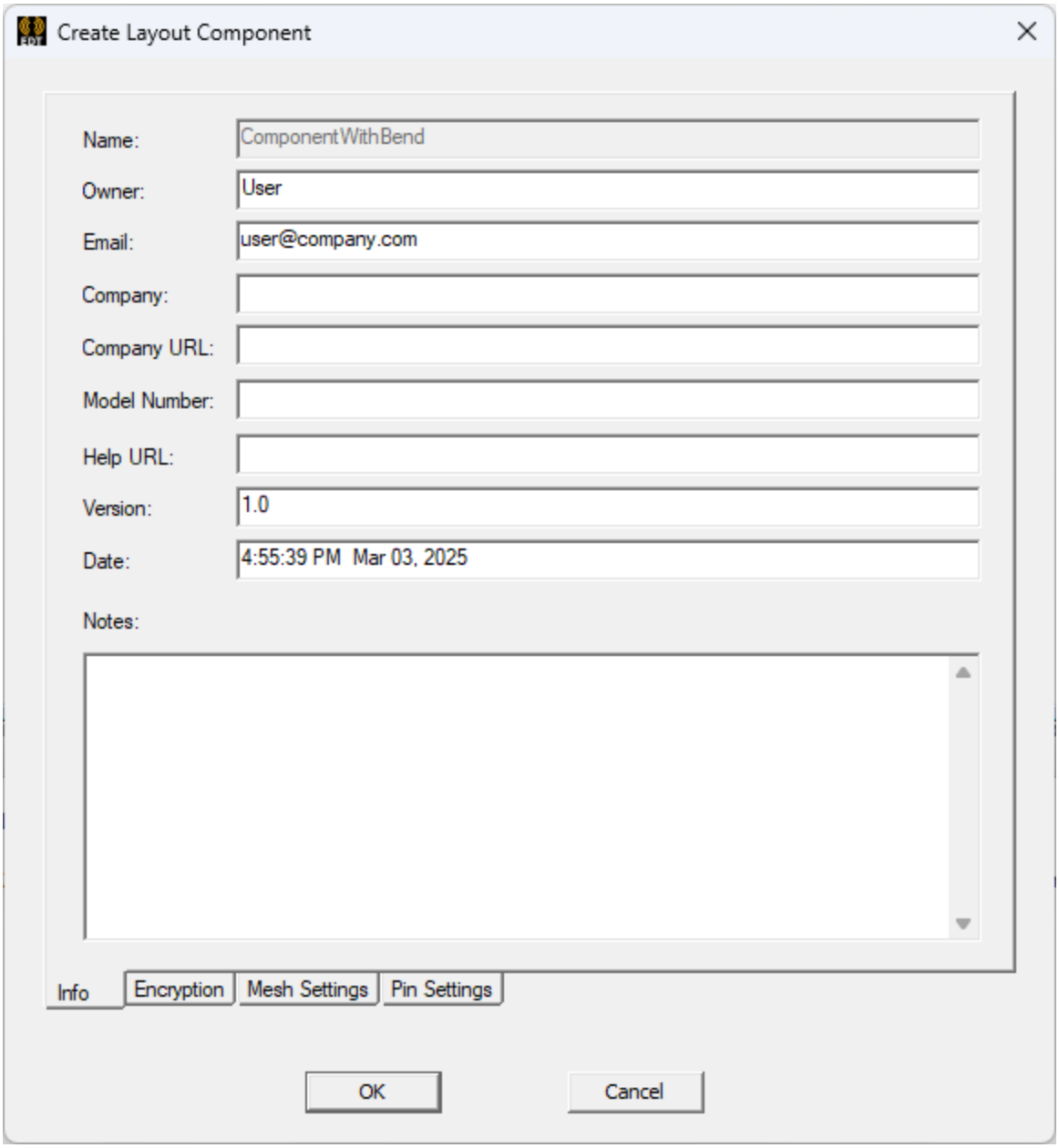
3. From the explorer window, enter a **File name** for the component (e.g., **ComponentWithBend.aedbcomp**). Then click **Save** to open the **Create Layout Component** window.

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



4. The **Create Layout Component** window allows users to add meta data, encryption, and mesh settings to an HFSS 3D Layout design component before exporting it. From the **Info** tab, enter any metadata in the **Owner**, **Email**, **Company**, **Company URL**, **Model Number**, **Help URL**, **Version**, **Date**, and **Notes** fields. The **Name** field automatically populates with the component's 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.

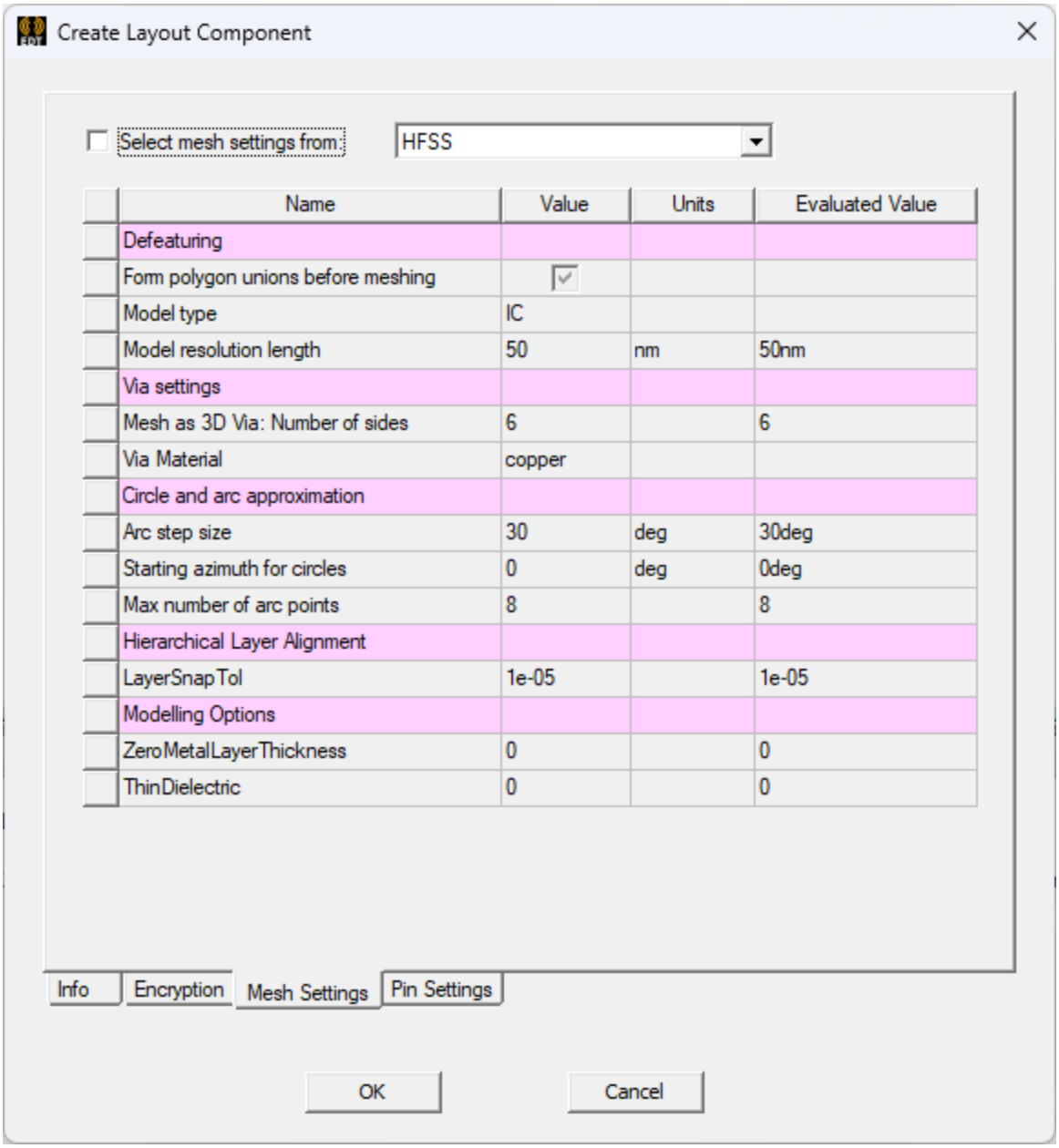


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The **Encryption** and **Pin Settings** tabs are accessible but does not function if the design contains a rigid-flex bend. Attempting to set up encryption results in the following error message.
>
> Refer to the Importing and Exporting Components for more information about the **Encryption** tab.

5. Navigate to the **Mesh Settings** tab. From the **Mesh Settings** tab to export solve settings on the original component. To do so, check the **Select mesh settings from** box and make a selection 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.



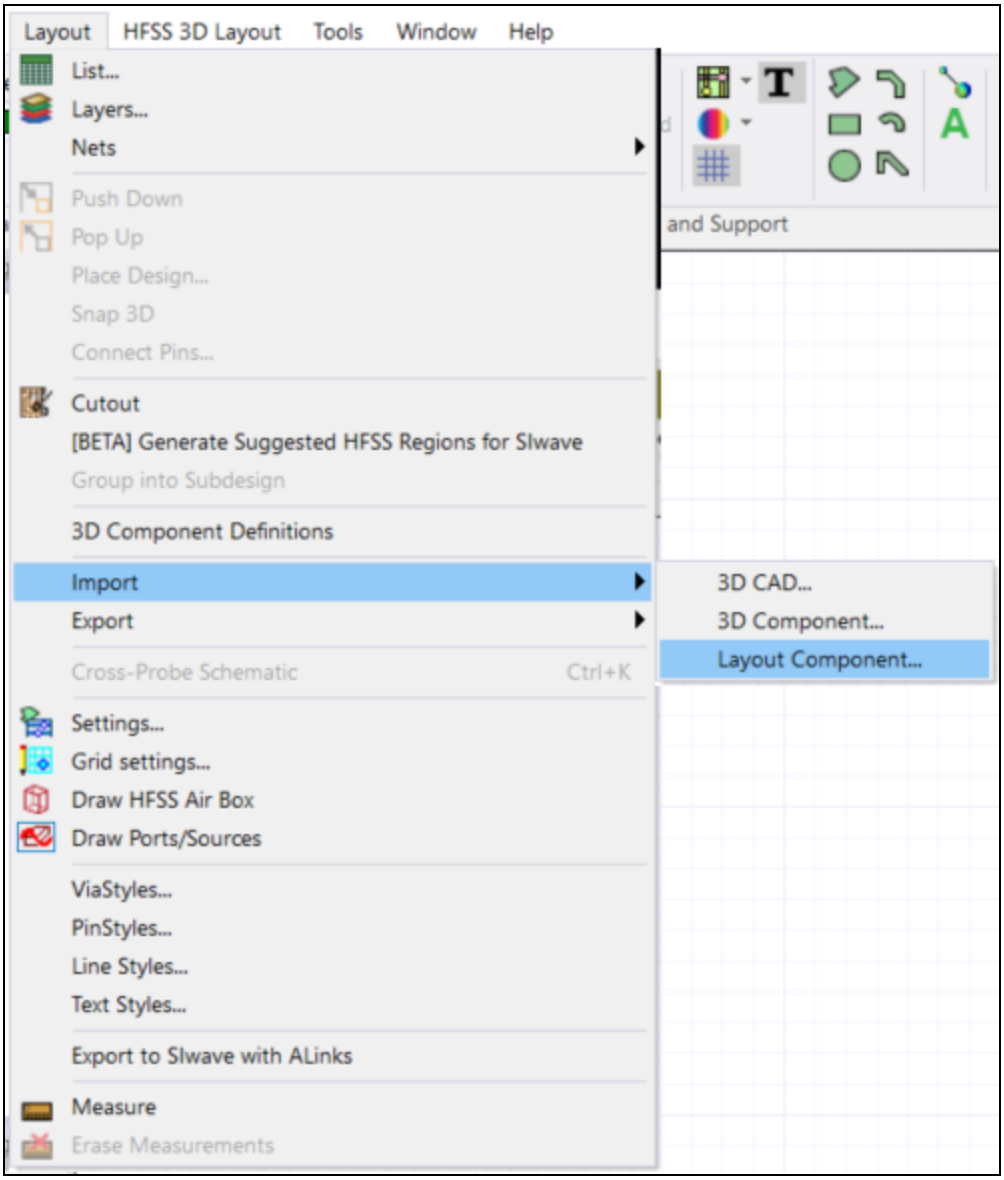
6. Click **OK** to export the component.

The procedure 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.

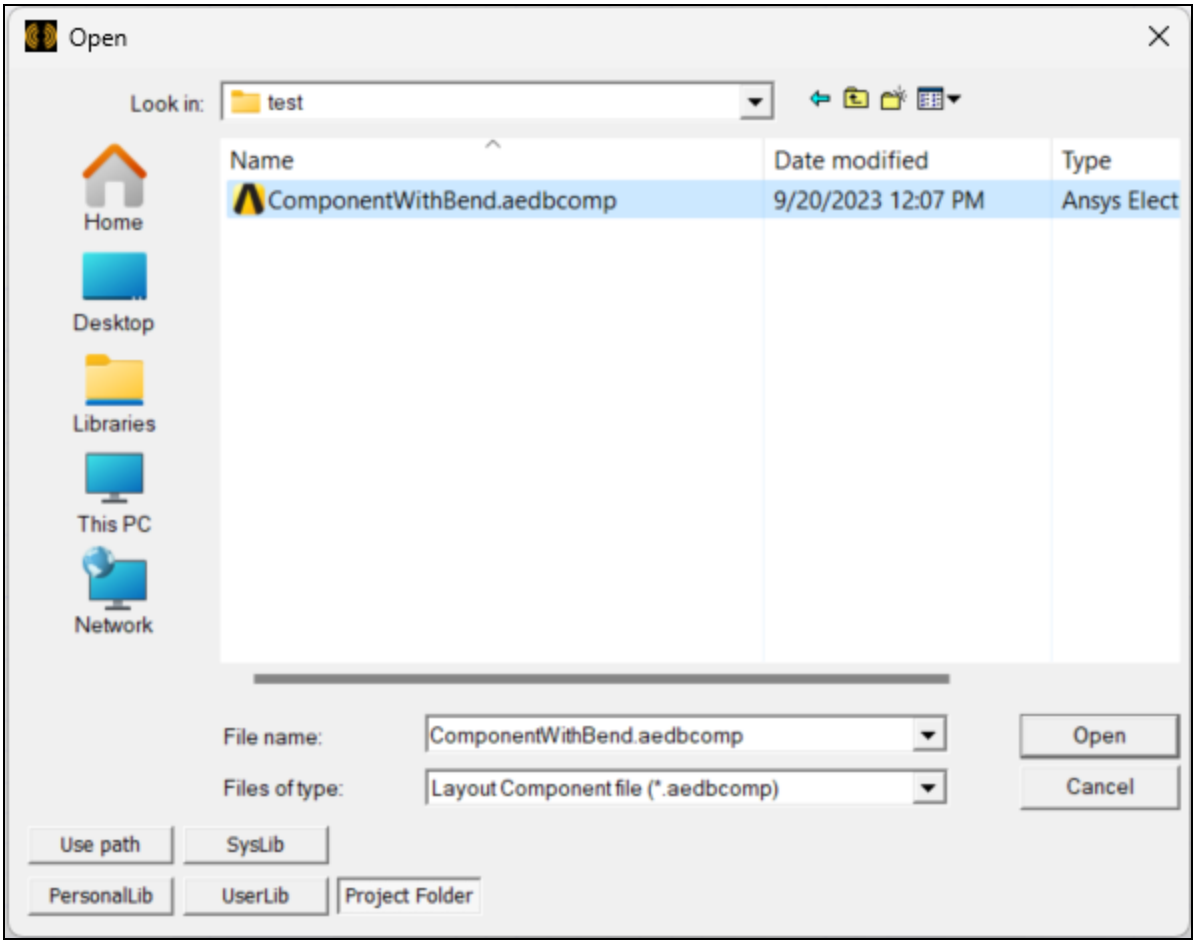
Importing a Flexed Design Component

1. Navigate to **Layout** and select **Import** > **Layout Component** to open an explorer window.

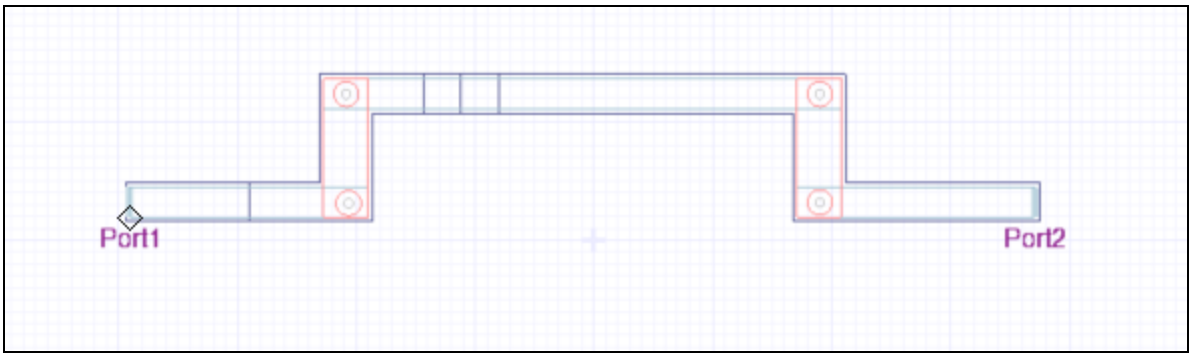


2. From the explorer window, navigate to an appropriate file (e.g., **ComponentWithBend.aedbcomp**). Select it, then click **Open** to import the component.

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

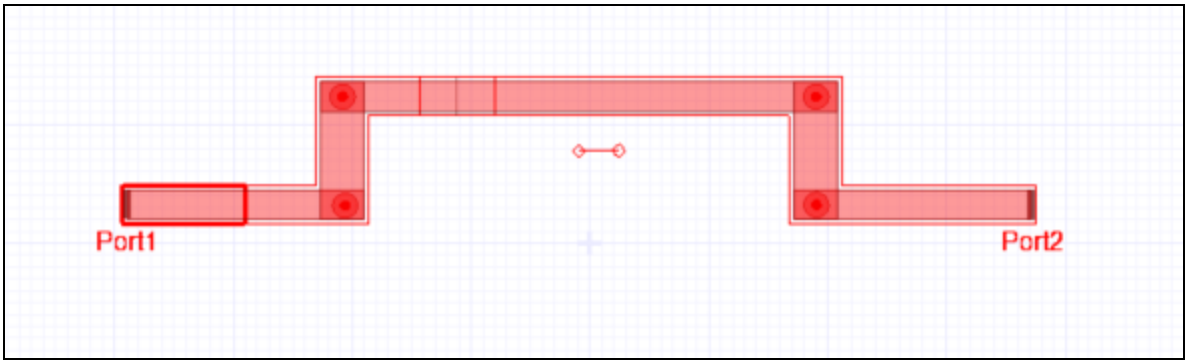


3. The new component appears in the **Layout Editor** attached to the cursor.

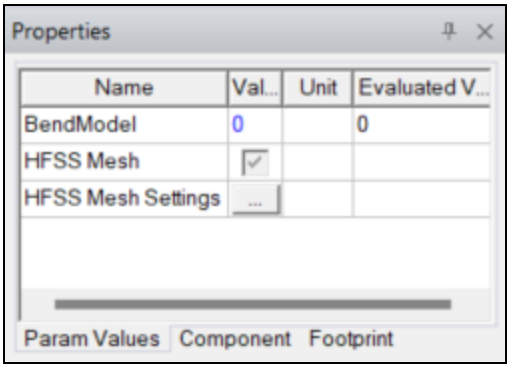


Navigate to an appropriate location to place the component, then click to set it.

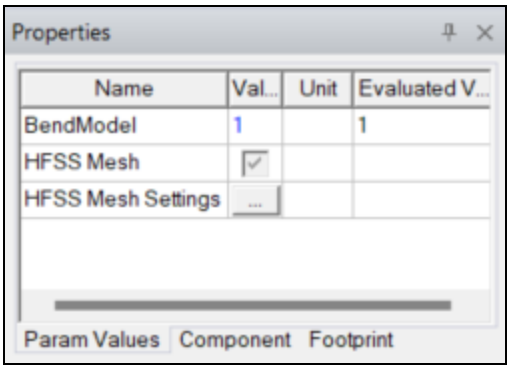
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. While the component is selected, navigate to the **Properties** window.

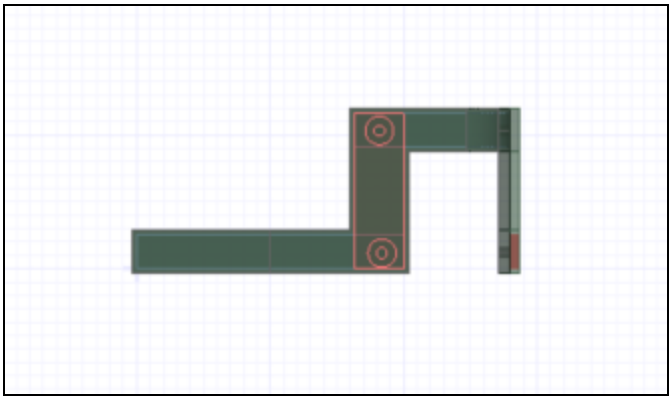


5. Enter **1** in the **BendModel** field.



Press **Enter** to apply the rigid-flex bend to 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.



6. Enter **0** in the **BendModel** field and press **Enter** to revert the component to its unbent state.
7. If applicable, make changes to the component. Refer to Editing a Component Definition.

The procedure is complete.

Overlapping Stackup

In overlapping mode, signal layers can exist in the same location (i.e.,overlap one another) on the Z-axis. Each layer also extends over the entire design space in the X-axis and Y-axis.

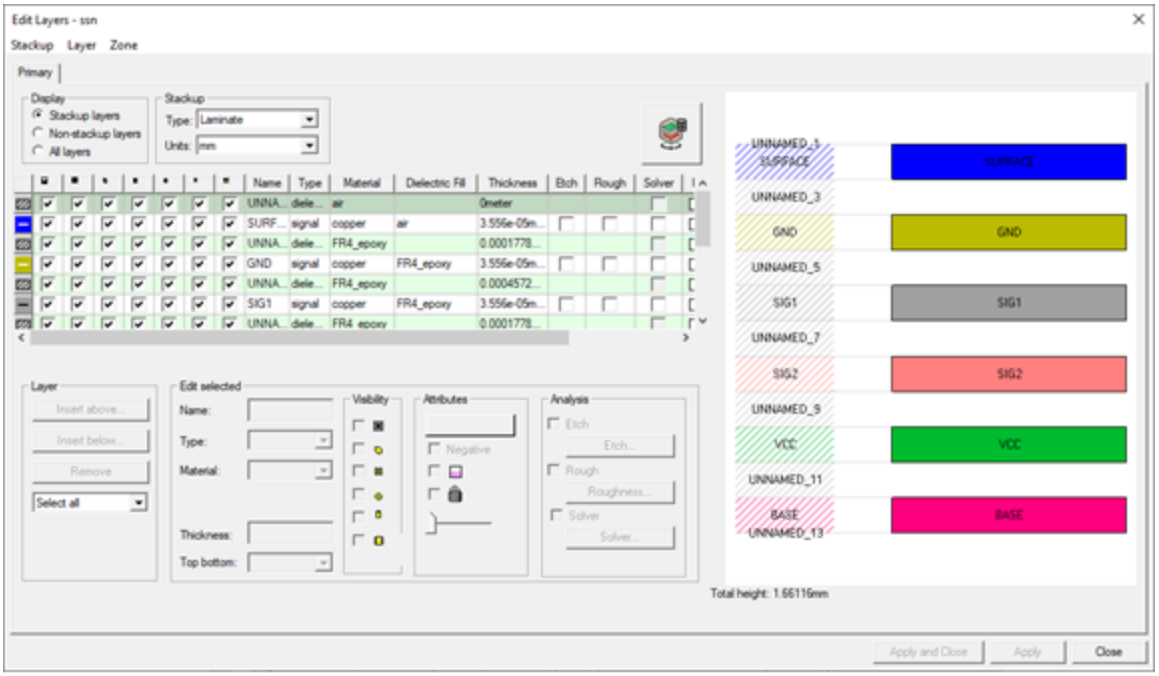
Dielectric layers must be contiguous (i.e., they cannot overlap nor have space between one another). Signal layers can reside at any elevation and can overlap dielectric and other signal layers. Via layers connect two signal layers. For rules on how layers may overlap, see Elevation in Overlapping Stackups.

When a new layer is created, the window automatically offers the same type of layer highlighted in the **Grid Control** pane.

Footprint Stackup

To access the footprint stackup, after the component has been placed in the **Layout Editor**, right-click the component and select **Edit Footprint** to open the **Footprint Editor**. Then, on the **Footprint** menu, select **Edit Layers**, which opens the Edit Layers 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 now use the Edit Layers window controls to insert or remove layers and configure the various layer attributes and configure stackup layers.

Model a Signal Layer as a Ground Plane

HFSS 3D Layout lets you model a signal layer as ground plane.

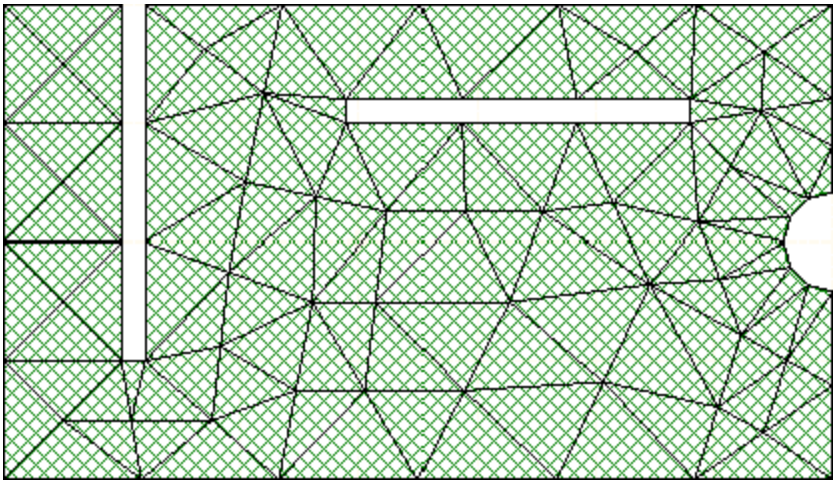
1. Open the **Edit Layers** window by:
   - On the menu bar, click **Layout** > **Layers** or **Footprint** > **Layers**.
   - In the **Layout** window, right click and chose **Edit Layers**.
   - On the **Layout** or **Footprint** ribbon, click the **Edit Layers** icon.
2. Add or select an existing signal layer.
3. From the **Analysis** area, select **Solver** to open the **Solver Options** window.
4. From the **Planer EM** tab, select **Treat as infinite ground** to model a signal as ground plane.
5. Click **OK** to close the **Solver Options** window.
6. Click **Apply** in the **Edit Layers** window.

The following diagrams illustrate the difference when a signal is modeled as a ground plane.

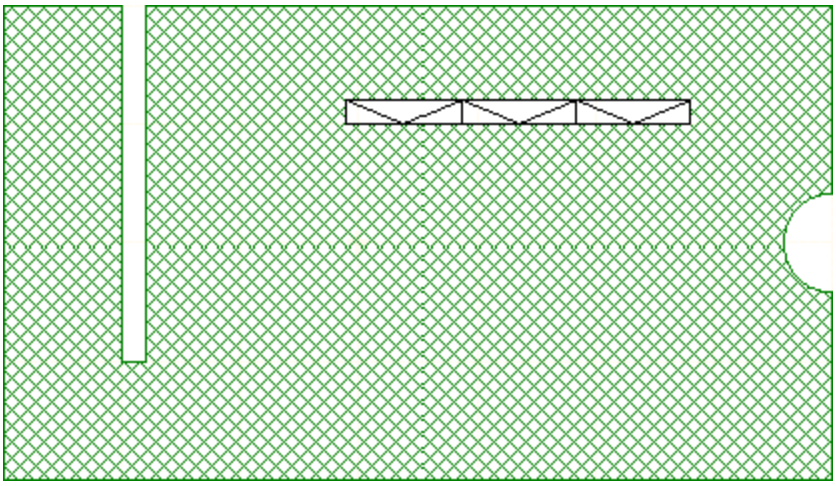
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When a signal layer is modeled as a ground plane, only the holes are modeled.

Modeled as a Signal Layer



Modeled as a Ground Plane



Subcircuits with Independent Stackups

Subcircuits can be pasted into an HFSS 3D Layout design as an instance of the original, with an independent stackup. The vertical placement of the sub-design is determined by the placement layer of the cell instance:

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 sub-design in the layout editor (without pushing down into it).
2. Set the placement layer use the “PlacementLayer” attribute.

- The HFSS 3D Layout ignores sub-designs with independent stackups during analysis; a warning is issued.
- For HFSS, a consistent mesh that includes all sub-designs is constructed. If negative signal layers intersect, the mesh fails to generate.
- HFSS 3D Layout attributes in the black-boxes is ignored (e.g., boundary conditions).
- Ports must be added at the analysis level; ports internal to any sub-design is ignored.
- Ports can be referenced to negative signals or metal on a signal layer in the sub-design containing the physical port location or any parent design.

Note:

- The stackup cannot be inverted; the lower elevation of the black-box stackup is always aligned with the top of the placement layer.
- The entire combined geometry is simulated by HFSS; any existing results for the independent sub-design are not re-used.

Using the Layer Stackup Wizard (HFSS 3D Layout)

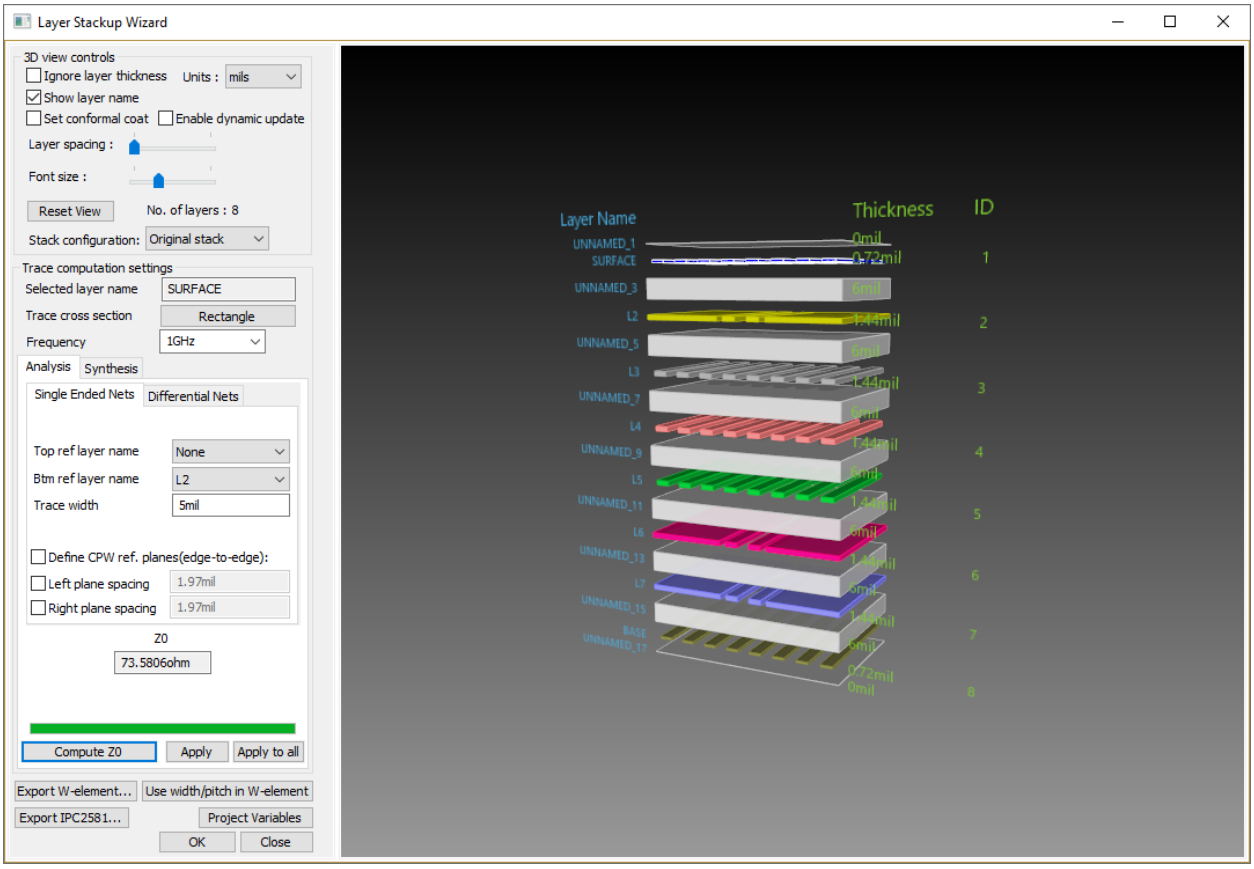
The Layer Stackup Wizard is a powerful utility that allows users to build and modify laminate layer stacks to meet precise impedance requirements for single-ended and differential microstrips and striplines. SIwave’s fast and accurate method-of-moments transmission line solver is used in all calculations and fully accounts for frequency-dependent dielectrics, non-uniform (e.g., trapezoidal) trace cross-sections, and surface roughness parameters. In Analysis mode, users can assess the impact of changing material parameters, layer thicknesses, etc., on the impedance of single-ended and differential lines whose width and separation are fixed. In Synthesis mode, users can compute trace width required to achieve a specified impedance value on a given layer. For differential nets, users can compute either trace width or trace separation to achieve a given impedance and trace separation (or width) values. Computed trace widths can then be applied to SIwave’s layout editor. Traces that are subsequently routed on layers with updated width values has the exact impedance value specified in the Stackup Wizard

A key component of this utility is the interactive, 3D visualization of the layer stack. This view, which can be panned, zoomed, and rotated using the mouse, provides users with an intuitive understanding of the dimensions and composition of their layer stack.

Opening the Layer Stackup Wizard

From **Layout**, select **Layers** to open the **Edit Layers** window. Then click the **Layer Stackup Wizard** icon to open the **Layer Stackup Wizard** window.

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



The **Layer Stackup Wizard** window contains two panes:

- **Control Pane** – located on the left, this pane allows you to configure the layer stack, and to calculate Impedance, Differential Impedance, and Trace Width/Pitch.
- **View Pane** – located on the right, this pane displays the current layer stack. The pane automatically updates as you change settings in the Control Pane, and you can add, remove, or edit certain layer properties directly from this pane. Use the mouse or hot keys to rotate and zoom the model. Use Ctrl+D to auto-zoom.

Configuring the Layer Stackup

From the **3D View Controls** area, make the selected selections on the following options.

- **Ignore layer thickness** – Checking this renders all layers at the same thickness. If this option is unchecked, layers display at their actual thicknesses.
- **Show layer name** – Checking this displays layer names. If this option is unchecked, layer material names display instead.
- **Set conformal coat** – Select this check box to add conformal coat layers. These layers are a set of dielectric layers on the top and bottom of all packages and PCBs 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 design. They have a default material of "SolderMask" and a default thickness of 15 microns. They are shown with a Layer Type of "CONFORMAL_COAT". If dielectric layers are already present above the top layer and below the bottom layer, their material is changed to SolderMask and the layer type is changed to Conformal Coat.

- **Units** – Use this drop-down menu to update the units displayed in the View Pane, and in the impedance calculator.
- **Enable Dynamic Update** – when enabled, dynamic update performs the following actions when any change is made in the Layer Stackup window:
  - The design updates.
  - Impedance and differential impedance are recalculated.
  - W-element plots are recalculated and replotted.
- **Layer spacing** – Use the slider to increase or decrease the space between layers.
- **Font size** – Use the slider to increase or decrease the size of the font in the View Pane.
- **Reset View** – Click to reset the View Pane to its default view.

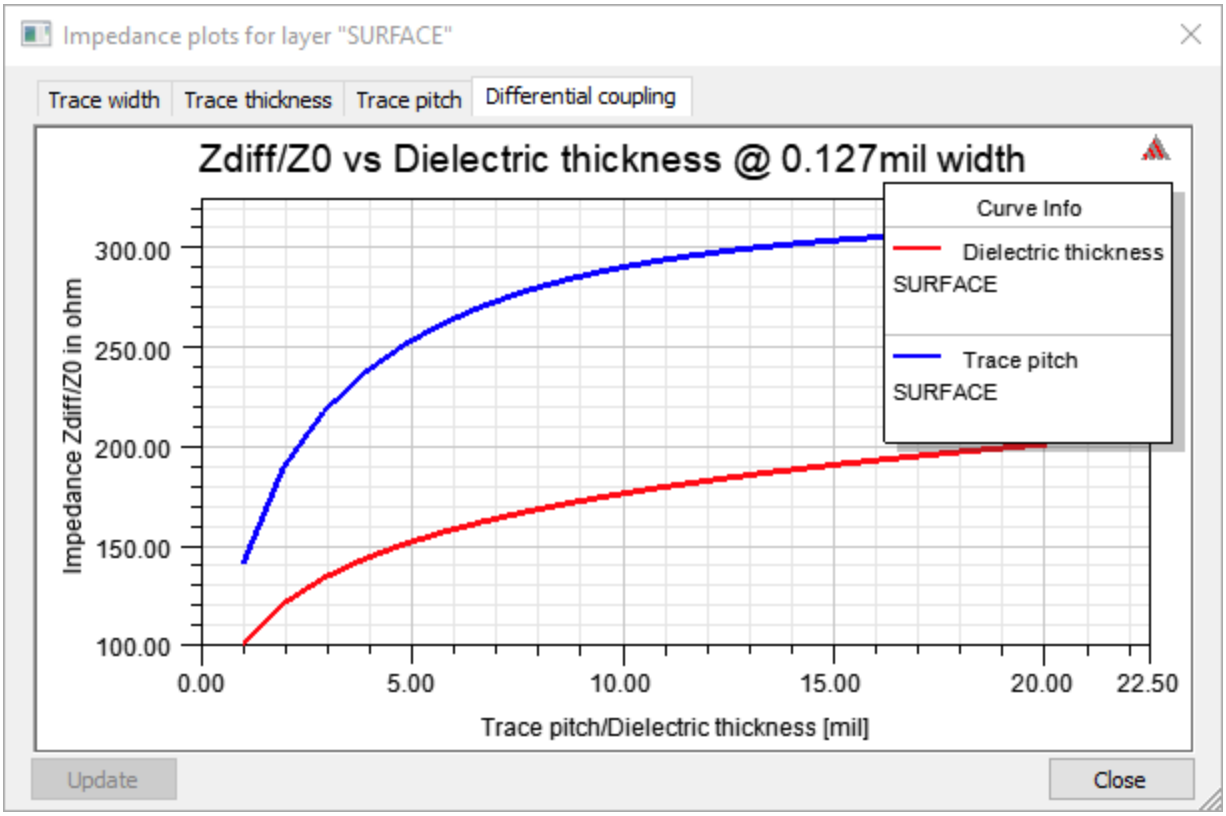
Exporting Data

- **Export IPC2581 File** – prompts you to select an *.xml or *.cvg file for export.
- **Export W-Element** – allows you to export the w-element.
- **Use width/pitch in W-element** – uses the calculated trace width or pitch in the w-element analysis.
- **Project Variables** – allows you to edit design variables.

Viewing Impedance Plots

Right-click a layer in the **View** tab and select **Show Impedance Plots** to open the **Impedance Plots for [Layer Name]** 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 plot is modelless. Make any change in the **Layer Stackup Wizard** to enable the **Update** button and use the tabs to navigate between plots.

Other Capabilities of the Layer Stackup Wizard

From the **Layer Stackup Wizard**, you can also:

- Calculate trace width or pitch
- Calculate impedance (Z0)
- Calculate differential impedance (Zdiff)
- Export W-element
- Compute insertion/return loss and time domain crosstalk

Calculating Trace Width and Pitch Using the Layer Stackup Wizard (HFSS 3D Layout)

From the **Trace Computation Settings** area of the Layer Stackup Wizard, you can configure and perform a trace width (or pitch) calculation.

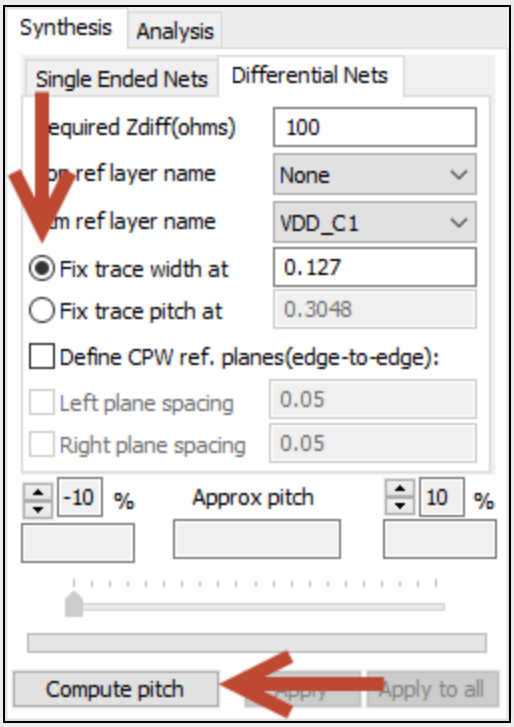
Prerequisites:

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 Pane to select a layer. The **Selected Layer Name** field displays the currently selected layer.
- Click **Trace Cross Section** to open the Trace Cross Section editor.
- Click the **Frequency** drop-down menu to select the frequency used for trace width calculations.

To calculate trace width or pitch:

1. Select the **Synthesis** tab.
2. Select the **Single Ended Nets** or **Differential Nets** tab.
3. Use the drop-down menus to select the required impedance or differential impedance (in ohms), and as reference layers.
4. If you are using the **Differential Nets** tab, select *either***Fix trace width at** or **Fix trace pitch at** and enter a value.

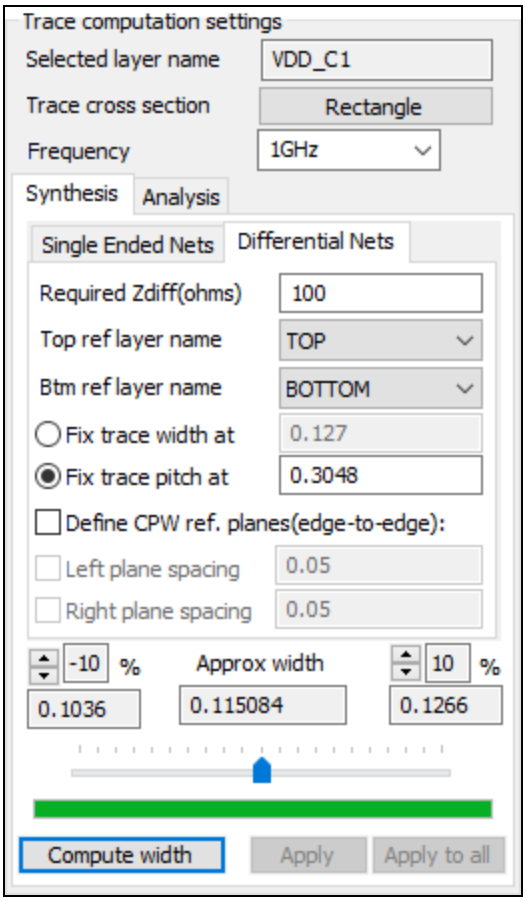
> **Important:**
>
> By default, **Fix Trace Pitch** is selected. Selecting **Fix Trace Width** changes the calculator so that it computes pitch instead of width.
>
> 
>

5. Select **Define CPW ref. planes (edge-to-edge)** to enable additional options (e.g., define coplanar waveguide reference planes).

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

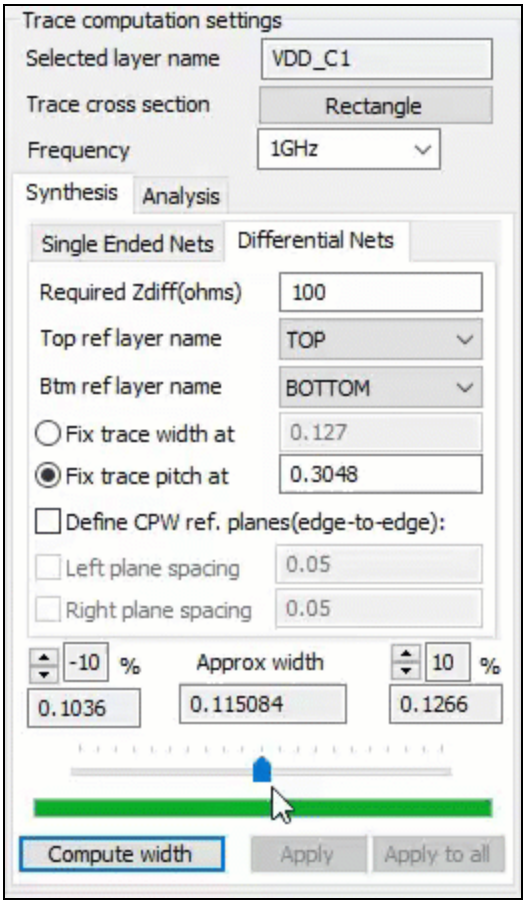
- You may select **Left plane spacing**, **Right plane spacing**, or both.
- For W-element export, selecting **Left plane spacing** means that aggressors may only be added to the right of a victim trace; any traces to the left are removed. Selecting **Right plane spacing** means that aggressors may only be added to the left of a victim trace; any traces to the right are removed.

6. To perform the calculation, click **Compute width**. The green progress bar shows the calculation's progress. When the calculation is complete, the **Approx width** area updates to show the calculated result.



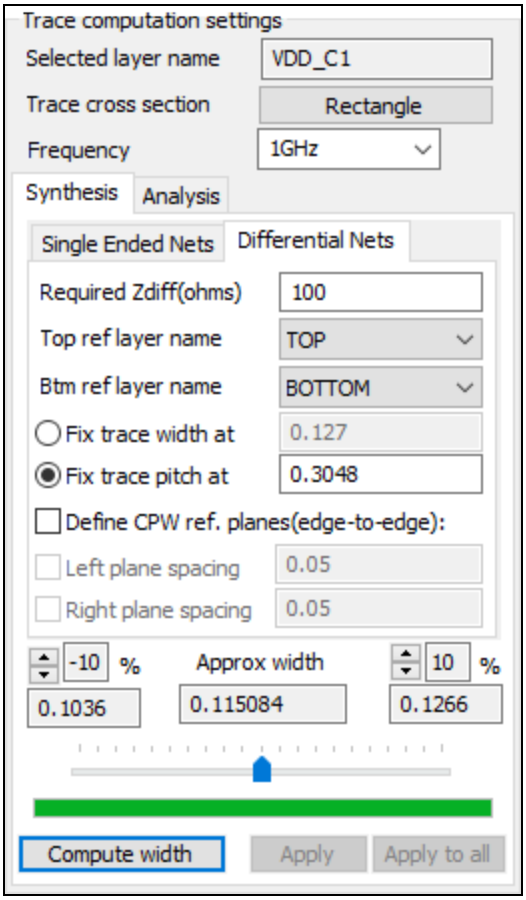
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Use the slider above the progress bar to update trace width and the required Zdiff value.



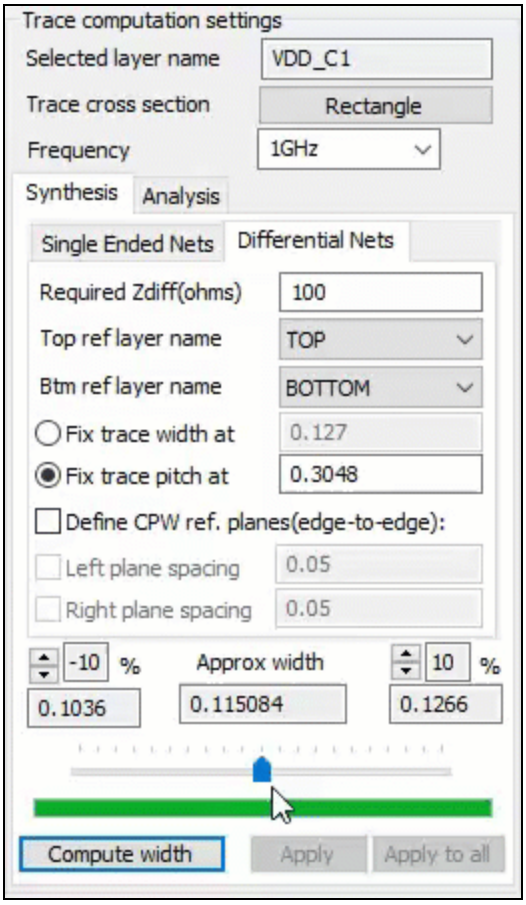
8. To perform the calculation, click **Compute width**.
9. The **Approx width** area updates to show the result.

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.

10. Use the slider above the green bar to update trace width and the required Zdiff value.



Calculating Impedance Using the Layer Stackup Wizard (HFSS 3D Layout)

From the **Trace Computation Settings** area of the Layer Stackup Wizard, you can configure and perform an impedance calculation.

Prerequisites:

- Use the View Pane to select a layer. The **Selected Layer Name** field displays the currently selected layer.
- Click **Trace Cross Section** to open the Trace Cross Section editor.
- Click the **Frequency** drop-down menu to select the frequency.

To calculate impedance:

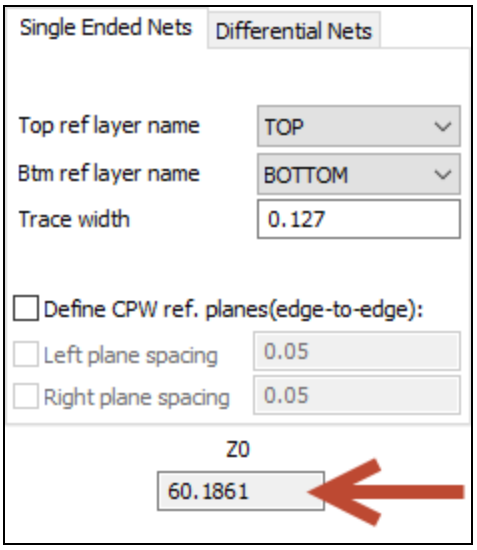
1. Select the **Analysis** tab.
2. Select the **Single Ended Nets** tab.

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 your reference layers.

> **Note:**
>
> Top and bottom reference layer must be selected in order to use multi-layer features of W-element export.

4. From the **Trace Width** field, enter a trace width.
5. Select **Define CPW ref. planes (edge-to-edge)** to enable additional options (e.g., define coplanar waveguide reference planes).
   - You may select **Left plane spacing**, **Right plane spacing**, or both.
   - For W-element export, selecting **Left plane spacing** means that aggressors may only be added to the right of a victim trace; any traces to the left are removed. Selecting **Right plane spacing** means that aggressors may only be added to the left of a victim trace; any traces to the right are removed.
6. Click **Compute Z0**.
7. The **Z0** field updates to show the result.



8. To apply this setting to one layer or all layers, click **Apply** or **Apply to all**.

Calculating Differential Impedance Using the Layer Stackup Wizard (HFSS 3D Layout)

From the **Trace Computation Settings** area of the Layer Stackup Wizard, you can configure and perform a differential impedance calculation.

Prerequisites:

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

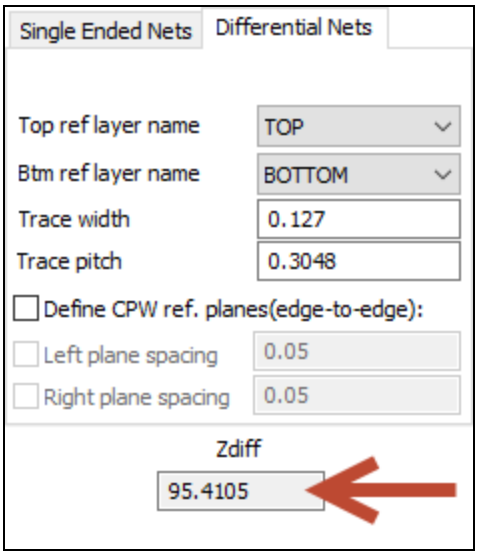
- Use the View Pane to select a layer. The **Selected Layer Name** field displays the currently selected layer.
- Click **Trace Cross Section** to open the Trace Cross Section editor.
- Click the **Frequency** drop-down menu to select the frequency.

To calculate differential impedance:

1. Select the **Analysis** tab.
2. Select the **Differential Nets** tab.
3. Select your reference layers.

> **Note:**
>
> Top and bottom reference layer must be selected in order to use multi-layer features of W-element export.

4. From the **Trace Width** and **Trace Pitch** fields, enter a trace width and pitch.
5. Select **Define CPW ref. planes (edge-to-edge)** to enable additional options (e.g., define coplanar waveguide reference planes).
   - You may select **Left plane spacing**, **Right plane spacing**, or both.
   - For W-element export, selecting **Left plane spacing** means that aggressors may only be added to the right of a victim trace; any traces to the left are removed. Selecting **Right plane spacing** means that aggressors may only be added to the left of a victim trace; any traces to the right are removed.
6. Click **Compute Zdiff**.
7. The **Zdiff** field updates to show the result.



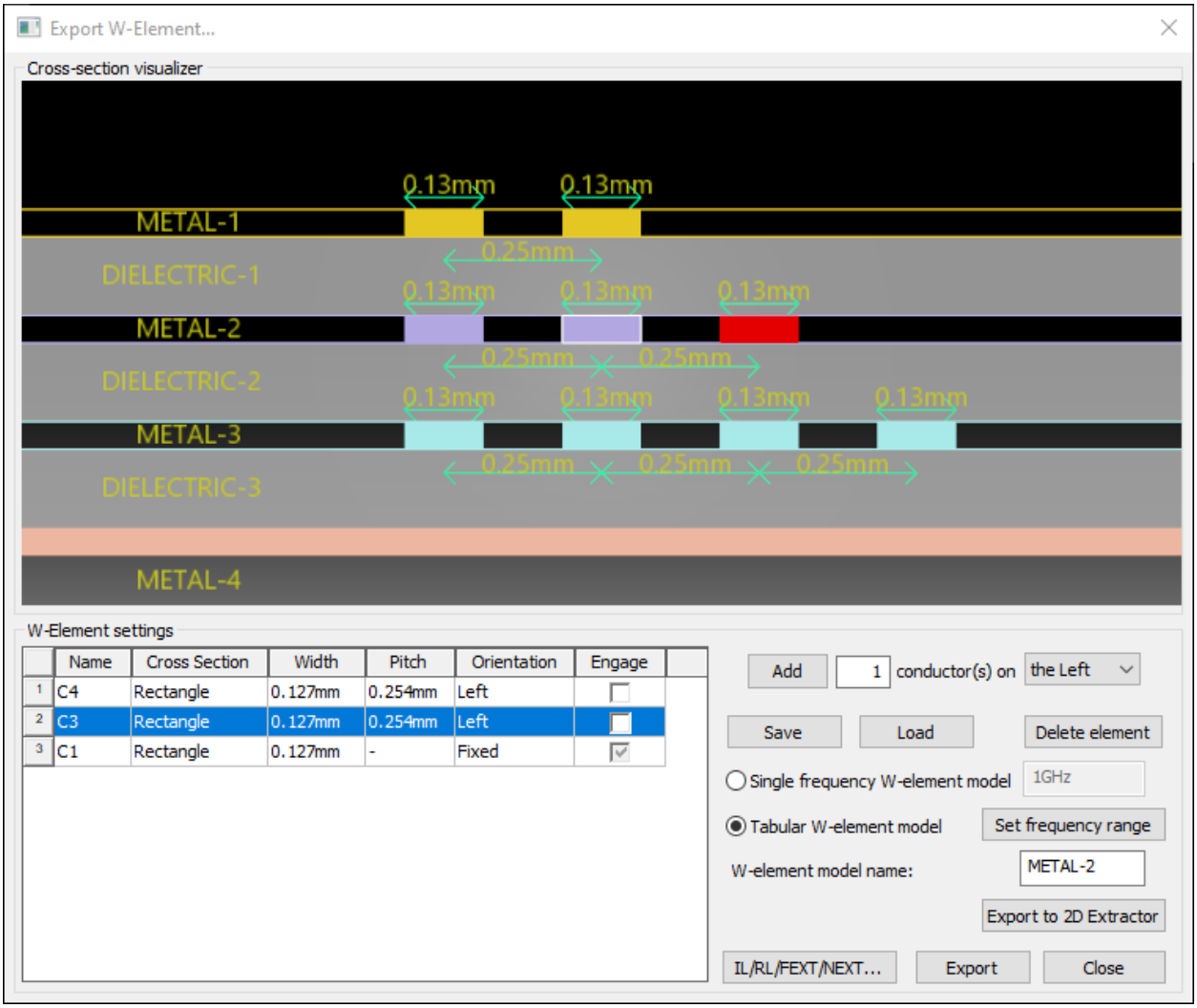
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting the W-element from Layer Stackup Wizard (HFSS 3D Layout)

After you have used the Layer Stackup Wizard to calculate impedance, differential impedance, or trace width/pitch, the **Export W-Element** window allows you to set up aggressor traces around a victim trace and export the corresponding w-element.

To configure a w-element export:

1. From the **Layer Stackup Wizard**, click **Export W-element** to open the **Export W-Element** window.



The **Cross-section Visualizer** displays a graphical representation of the **W-Element Settings**, including dielectric blocks and the top and bottom reference 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:**
>
> Also access this window by drawing a cross-section for w-element analysis.

2. Click a conductor in the visualizer or select it from the list in the **W-Element Settings** area.

   The grid displays the following for each conductor:

   - **Name** – conductor name.
   - **Cross Section** – indicates the shape of the cross section.
   - **Width** – can be edited here directly, or imported on the Layer Stackup Wizard by selecting **Use width/pitch in W-element**.
   - **Pitch** – can be edited here directly, or imported on the Layer Stackup Wizard by selecting **Use width/pitch in W-element**.
   - **Orientation** – fixed, left, or right.
   - **Engage** – enabling this check box engages the conductor in IL/RL/FEXT/NEXT simulations.

   The following options are available:

   - **Add** – Enter the number of aggressors to add, and use the drop-down menu to select whether to add them on the **right**, **left**, or on **both sides**. Then click **Add**.

     > **Note:**
     >
     > If you previously enabled CPW settings during trace width, impedance, or differential impedance calculation, your options will be limited.
     >
     > - If you selected **Left plane spacing**, aggressors may only be added to the right of a victim trace.
     > - If you selected **Right plane spacing**, aggressors may only be added to the left of a victim trace.

   - **Delete Element** – Click to remove an element.
   - **Single Frequency / Tabular Model** – Select **Single frequency** and enter the frequency in Hz, or select **Tabular** and click **Set frequency range** to set up a range of frequencies.
   - **W-Element Model Name** – If selected, enter a new model name for the w-element.
   - **Export to 2D Extractor** – generates a script to export the selected layer's cross-section to 2D Extractor.

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

The layer thickness, conductor width, pitch, etching, conformal coat thickness are all defined using design variables.

Consult the Q3D Extractor/2D Extractor help for instructions on running an analysis in 2D Extractor.

3. Click **IL/RL/FEXT/NEXT** to compute insertion/return loss and time domain crosstalk.
4. Make then selected selections, then click **Export** to open the **Select Spice File to Export** window.

> **Note:**
>
> Selecting **Tabular W-element model** but not specifying the frequency range will automatically open the **Frequency Sweep** window. You must specify the frequency range before you can save.

5. Browse to a save location, name your file, and click **Save**. The file is saved in Spice (*.sp) format.

Computing Insertion/Return Loss and Time Domain Crosstalk (HFSS 3D Layout)

From the **Export W-Element** window, you can compute and plot insertion loss (IL), return loss (RL), far-end time domain crosstalk (FEXT), or near-end time domain crosstalk (NEXT).

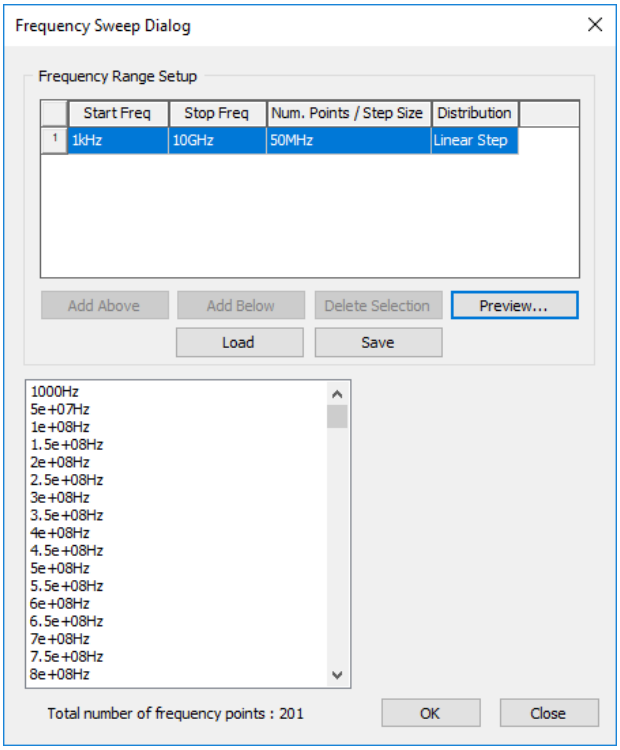
Prerequisites:

- Use the Layer Stackup Wizard to calculate trace width/pitch, impedance, or differential impedance.
- Export W-element.

To calculate and plot IL/RL/FEXT/NEXT:

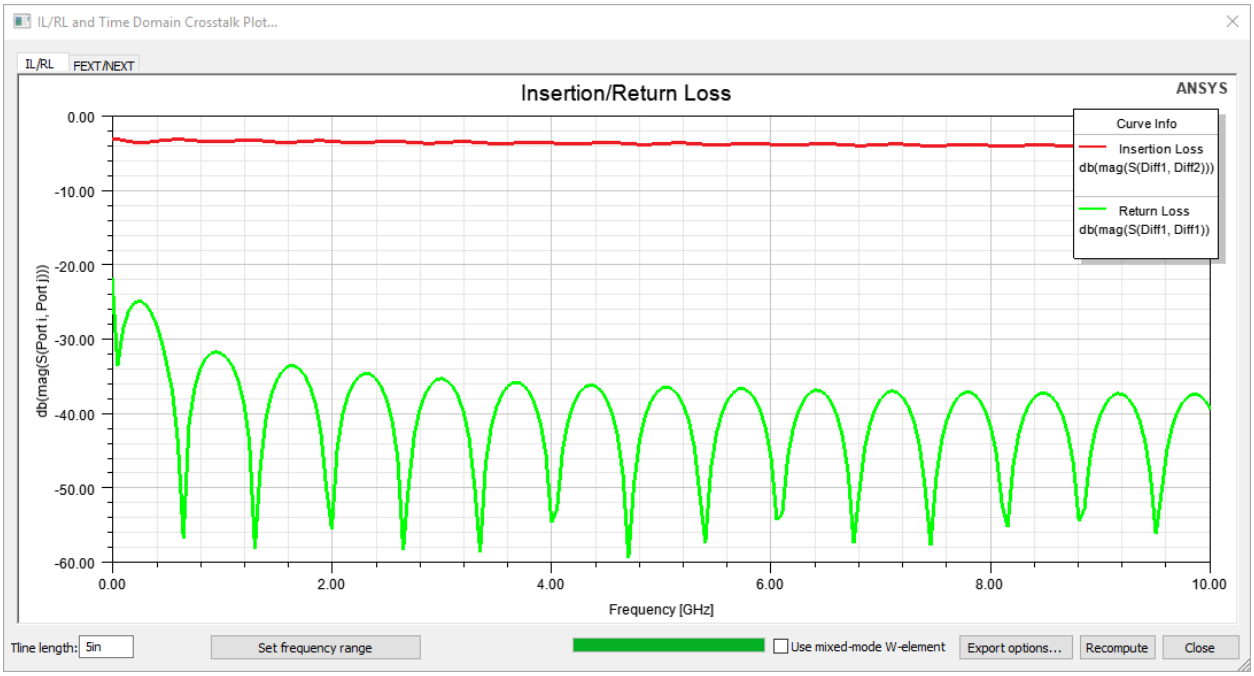
1. From the **Export W-Element** window, click **IL/RL/FEXT/NEXT** to open the **Frequency Sweep** window.

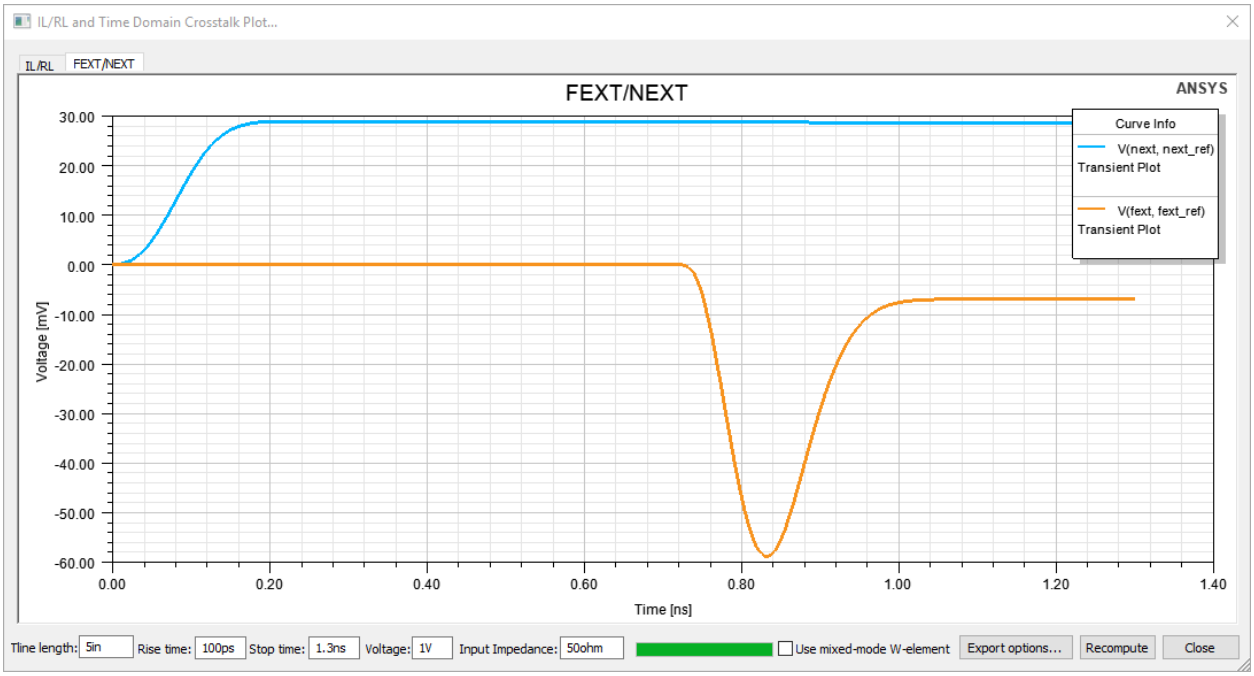
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Add, remove, and edit frequencies as appropriate. Click **Preview** to preview the number of frequency points.
3. Click **OK** to open the **IL/RL and Time Domain Crosstalk Plot** window to the **L/RL** tab.
4. Select the **FEXT/NEXT** tab to view the crosstalk plot instead. While the plot computes, a green progress bar displays the progress. When it is finished, the selected 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.





For IL/RL plots, you can adjust the **Length** or click **Set frequency range** to change the frequency range of 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.

For FEXT/NEXT plots, you can adjust the **Tline length** (trace length); edit the **Rise time**, **Stop time**, and peak **Voltage** of the driver attached to the aggressor net; or modify the driver **Input Impedance** value. This impedance is also used to terminate all transmission lines in the specified cross-section.

Click **Export Options**: to export plot data:

- For IL/RL plots, select **Export w-element schematic to circuit** (in *.aedt format) or **Export touchstone file**.
- For FEXT/NEXT plots, select **Export w-element schematic to circuit** (in *.aedt format) or **Export .csv file**.

If you change values and need to regenerate the plot, click **Recompute**.

Right-clicking in the plot enables additional plotting actions.

5. Click **Close** to exit the plot window.

Adding Programmatic EDB Cells to HFSS 3D Layout

1. From the **Project Manager** window, expand **Definitions** and right-click **Components**.
2. Choose **Add Scripted Component**.
3. From the explorer window, click one or more Python script files (.py) to generate a set of component, footprint, and symbol definitions.

> **Note:**
>
> Each Python script file must possess two elements:
>
> - A component, “MyNewComponent”, that is derived from IScriptedComponent. The IScriptedComponent file is located in the executables directory in PythonFiles\HFSS3DLayoutDesign\ScriptedComponent\IScriptedComponent.py either within Win64 or Linux64.
> - A function similar to the following that returns an instance of the new component class:
>
> def GetComponent():
>
> return MyNewComponent()
>
> Issues that arise upon import are reported through the Message Manager.

Creating a Component

Define all functions that return “NotImplementedError”.

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

- **GetVersion**: Should return a floating point number.
- **GetParameters**: Returns a list of Property objects (defined in the IScriptedComponent.py folder).
- **Property**: Is in the list returned by GetParameters.
- **MenuProperty**: Is in the list returned by GetParameters. MenuProperty expects a list of values as its argument list — not a list of strings. These are the default values used to generate the footprint, component and symbol. These values can only be edited at the Component Instance level. MenuProperty is a class derived from "Property". Both "Property" and "MenuProperty" are in the list returned by GetParameters.
- **GetTerminalNames**: Returns a list of terminal names that are used to establish the terminal order for the resulting component.
- **CreateMaterialDefinitions**: Must be defined even if the plan is to use the default definitions. Materials are created at the project level.
- **CreatePadstackDefinitions**: Must be defined even if it does nothing but “return”. Padstack definitions are added at the project level.
- **GetLayerList**: Should return a list of StackupLayers in order from top to bottom.
- **CreateFootprint**: where the data is available to use. Add geometry, padstacks, ports, etc. The ports should match up with the terminal list. The behavior is not well defined if that is not the case.

Editing the Python Script

Under **Footprints**, double-click or right-click to edit the footprint. This modifies the main menu. Then select **Footprint** and **Edit Script**. This allows modification of the Python script that is saved in the project. When finished making edits, select “Save/Run Script” under “Script” and all elements of the component are regenerated, as are component instances. Scripting issues are reported through the Message Manager.

Saving to Libraries

EDB Scripted Components can be saved just like any other component, footprint, or symbol. However, hand editing these libraries is not recommended.

Component Instances

Just like when any parametrized component is edited, modifying the parameters of the component are reflected in the component instance. If the “VerifyParameters” function is overloaded, any invalid parameter results in the parameter reverting to its previous state. This are reported through the Message Manager.

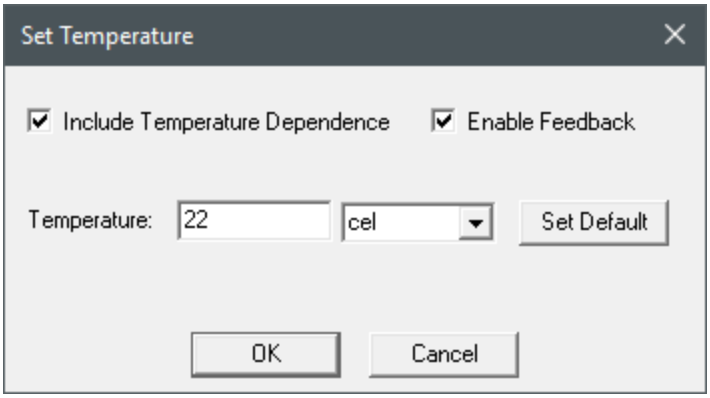
Setting Uniform Temperature in HFSS 3D Layout

To set a uniform temperature and enable feedback for use in two-way coupling:

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 the **HFSS 3D Layout** > **Set Temperature** command to display the **Set Temperature** window, or right-click the design in the **Project Tree** and select.

   This window allows you to set a global design temperature, which is used to modify the conductivity of metallization during the DC IR simulation. In general, the higher the temperature, the lower the conductivity (implying a higher resistance).



2. To enable the editing features, check **Include Temperature Dependence**.
3. Enter a value for Temperature, or an existing variable in the field.
4. Make a selection on the units drop-down menu.
5. If needed, **Set Default** to make the specified values the default.
6. To enable coupling with thermal designs, select **Enable Feedback**.

   You can perform two-way coupling between an HFSS 3D Layout design with an Ansys SIwave™ DC IR solution setup and an Icepak design in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. In an Icepak design, you can specify the number of coupling iterations to be run. DC IR 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.

7. To close the window and accept the changes, 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.

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

7 - Layout Editor

The **Layout Editor** shows a physical representation of a circuit. Layers are used in the editor to organize and manipulate sets of geometry and other visual indicators.

- Signal, negative signal, and dielectric are common physical layers.
- Symbol, error, and rats are non-physical layers.

The **Edit Layers** window 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.

IC Mode

The Layout Editor opens in either General mode (i.e., default) or IC mode. IC mode integrates ECAD Xplorer into the HFSS 3D Layout environment, preserving ECAD Xplorer's capabilities while allowing users direct access to most General mode, such as:

- HFSS 3D Model Extents
- Meshing
- Clip Planes
- Post Processing and Generating Reports
- Recording a Script to a File
- Parametrics
- Components (except for 3D Components)

In other circumstances, ECAD Xplorer features have been expanded to include HFSS 3D Layout capabilities, such as in the Edit Components window.

Finally, all of ECAD Xplorer's native features have been integrated and are accessible through shared topics, as well as through the following IC Mode-specific topics:

- Creating Circuit Ports in IC Mode
- Adding a Solution Setup
- Uniting Primitives in IC Mode

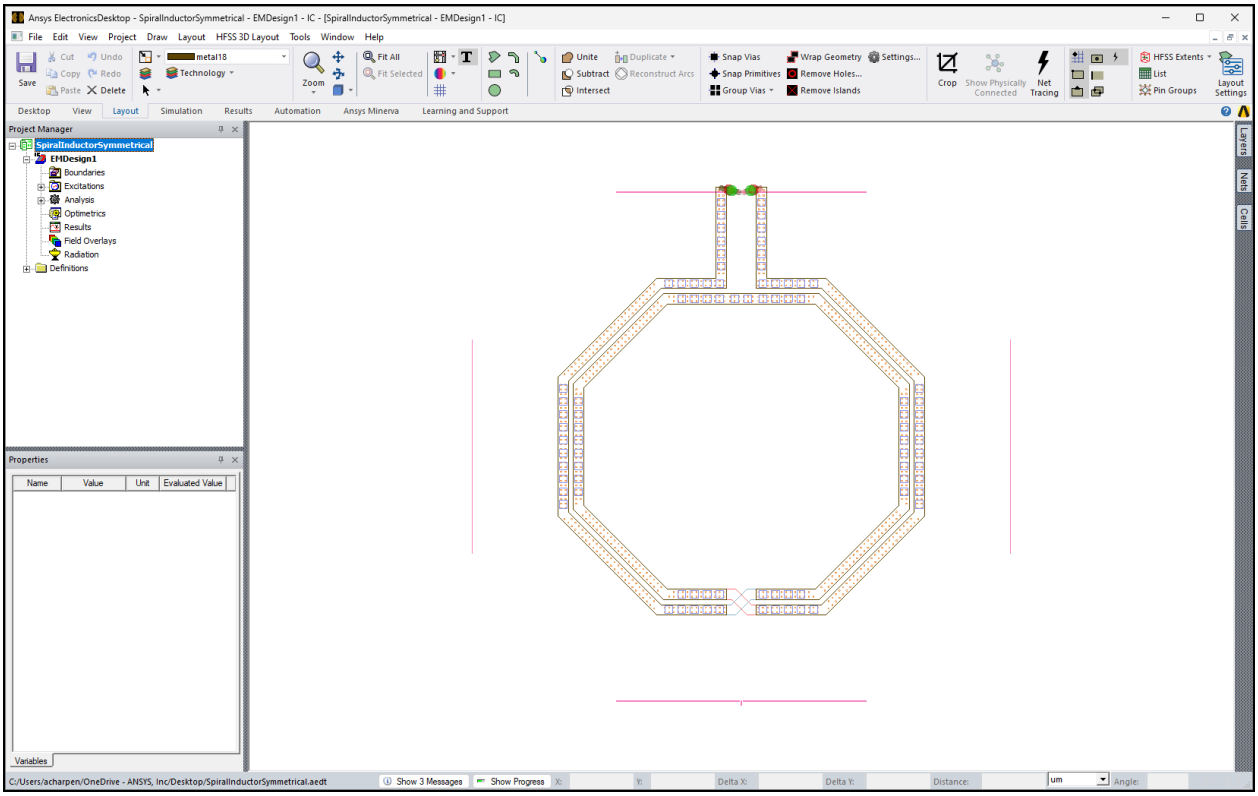
Refer to Switching to IC Mode.

IC Mode Workflow

IC mode typically uses this workflow:

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

1. Import a GDSII file with a control file.
2. Simplify the design using the tools available in IC mode:
    a. Crop and/or delete inappropriate objects.
    b. Group vias.
    c. Snap vias.
    d. Wrap geometry.
    e. Snap primitives.
    f. Trace nets.



Switching to IC Mode

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform will automatically reconfigure the HFSS 3D Layout ribbon and surrounding windows for the IC mode layout environment whenever a GDSII file is imported. Users can also switch between General and IC mode from either the **HFSS 3D Layout** menu or the active design'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.

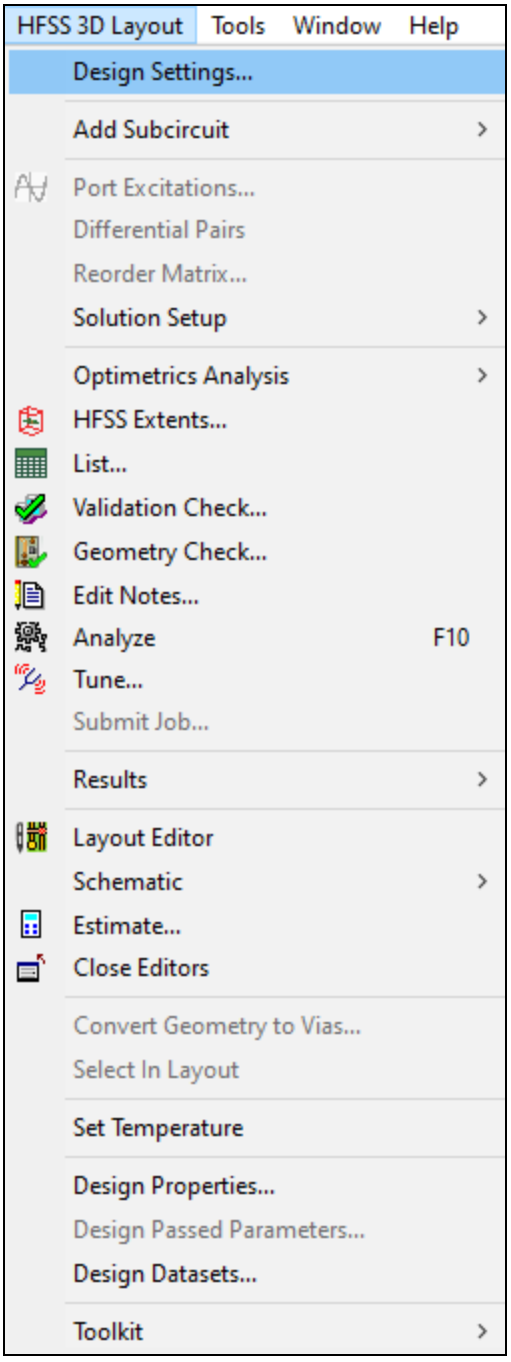
Note:

If users attempt to open a design in IC mode that fails to meet the required parameters, or includes hierarchy that IC mode cannot interpret, the design remains active in General mode only. Refer to Opening a Design in IC Mode.

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

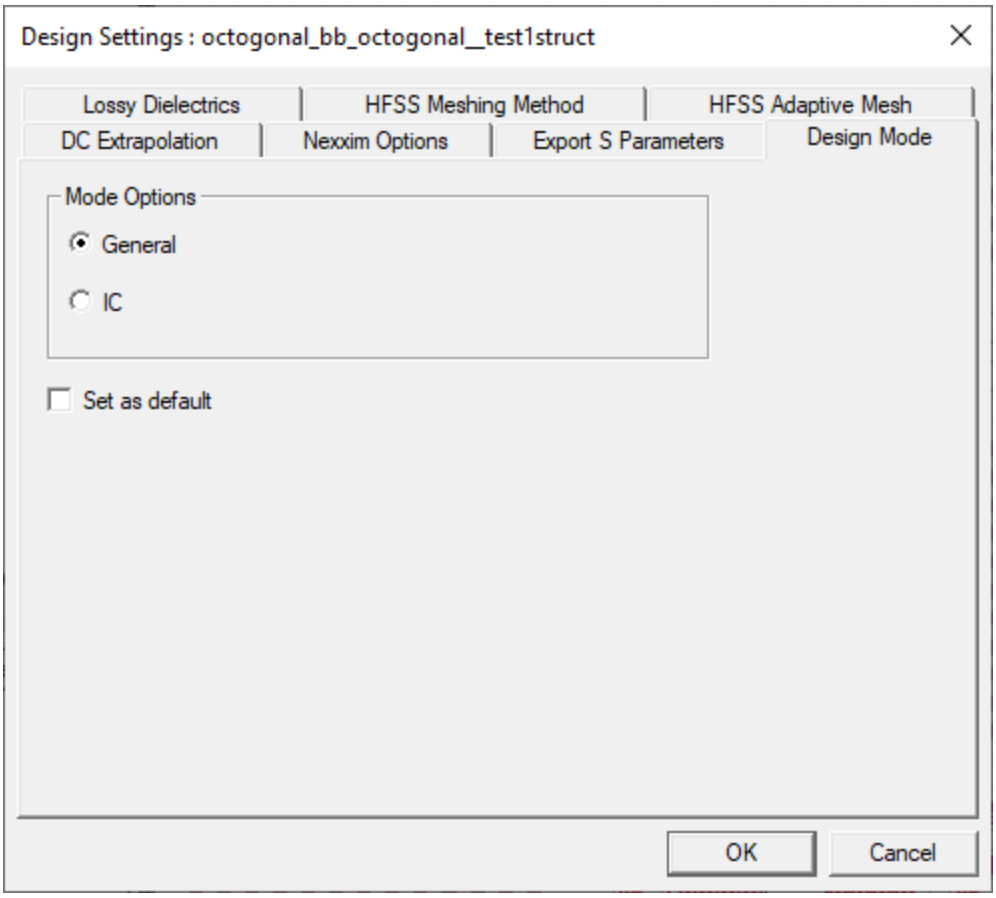
Switching to IC Mode from the HFSS 3D Layout Menu

1. Navigate to **HFSS 3D Layout** > **Design Settings** to open the **Design Settings** window.



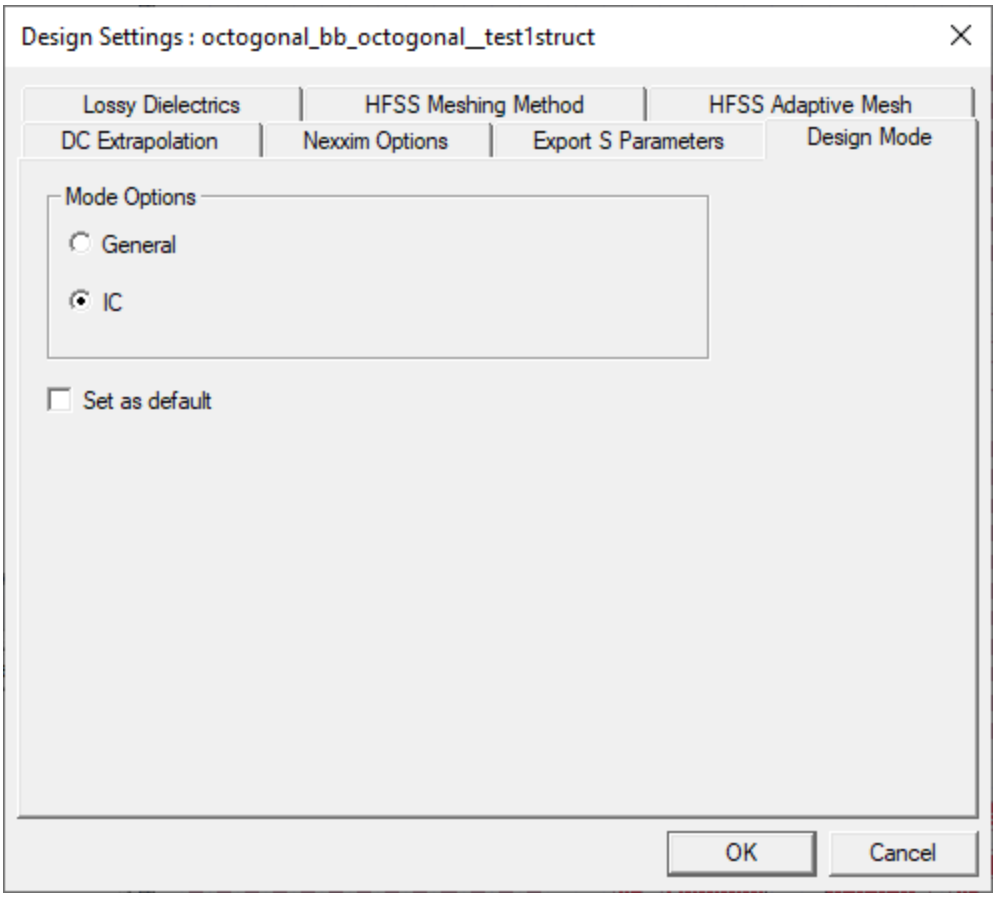
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Navigate to the **Design Mode** 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 **IC** from the **Mode Options** area.

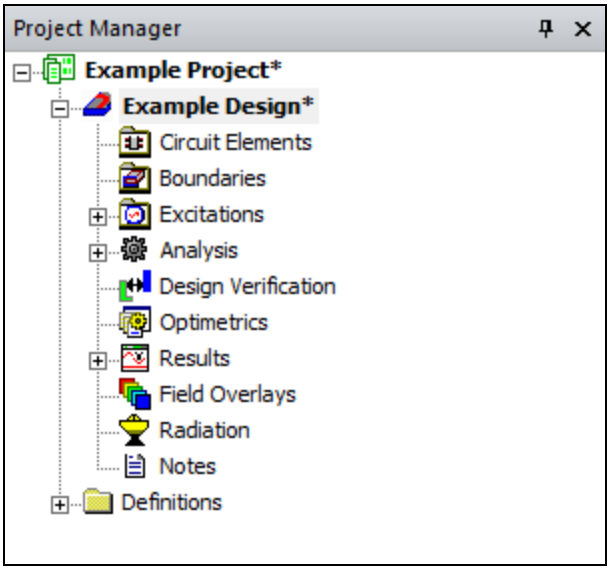


4. Click **OK** to close the **Design Settings** window.
5. The ribbon and windows will reconfigure to reflect the selected mode. Users will also see their selection reflected in the **Project Manager**. As long as a design is in IC mode, the words "IC Mode" will appear in parenthetical citation adjacent to the design icon (e.g., **Example Design**).

Switching to IC Mode from the Properties Window

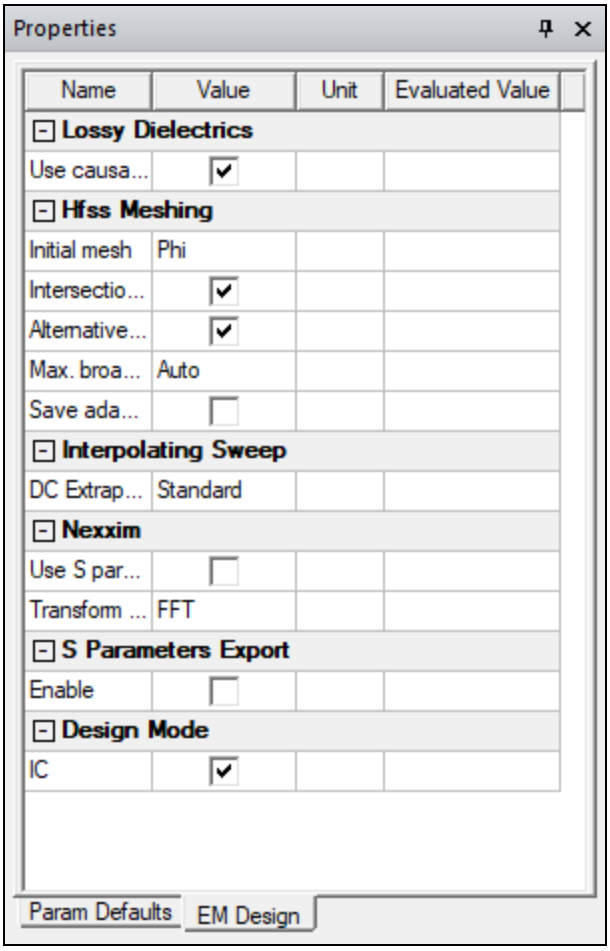
1. From the **Project Manager**, expand the **Project Tree** and select the **[*Active Design Folder*]** (i.e., **Example Design**) to populate the Properties window with the design's parameters.

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



2. Check the box in the **Design Mode** > IC mode.

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



3. The ribbon and windows will reconfigure to reflect the selected mode. Users will also see their selection reflected in the **Project Manager**. As long as a design is in IC mode, the words "IC Mode" will appear in parenthetical citation adjacent to the design icon (e.g., **Example Design**).

Opening a Design in IC Mode

The following lists explain why a design may be inactive in IC mode or General mode:

Inactive in IC mode if

- GDSII file is imported
- Design contains hierarchy
- Design contains a multizone stackup

Inactive in General mode if

- Design includes hierarchy
- Design contains encrypted technology

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

- Design is a symbolic footprint
- Design is a component
- Design contains a component
- Design contains a wave port
- Design contains a hierarchical port.
- Design contains a padtack instance across layers
- Design contains a negative primitive
- Design contains a RaptorX setup
- Design contains a Q3D setup

Layout Editor Display and Menus

The following topics describe the display, menus and options that are available for viewing and editing in the Layout Editor.

Layout Editor Display

The default display of the **Layout Editor** is 3D, with a view in a top-down 2D Cartesian coordinate orientation. The view can be changed by holding the left mouse key, dragging the cursor, in combination with the following keystroke(s):
Shift — **PanShift**+**Alt** — **Zoom**

Double-click while pressing **Shift**+**Alt** performs a fit-all operation.
Alt — **Rotate** Double-click while pressing **Alt** snaps the view to various predefined orientations.
Double-clicking in the screen region at:

- Top-left, top-right, bottom-left, or bottom-right — **orientates the view obliquely**.
- Right-center — orientates the view down the positive Y axis; positive X is to the left (**side-on view**).
- Left-center — orientates the view down the negative Y axis; positive Y is to the right (**side-on view**).
- Center-top — orientates the view down the positive Z axis with the X axis facing up (**top-down view**).
- Center-bottom — the resulting view looks down the negative Z axis with the X axis facing down (**bottom-up view**).
- Center — **resets the view to a standard 2D Cartesian orientation** with X to the right and Y up.

All object drawing occurs in the plane of the layer. Object manipulation, such as dragging, occurs in the plane of the object. Drawing in an oblique view places 2D objects on the active

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

layer, even if a snap occurs to a point that is off-layer. The background grid is drawn at the layer elevation, unless the view is too horizontal. To select from several vertically layered objects, use the B key to cycle through all the objects that exist directly beneath the cursor.

Layout Editor Options

Layout options control the display, drawing, and editing of models in the Layout Editor. All layout options have installed default values. Option values may be changed on their installed defaults either globally (for all new layouts) or locally (for a particular existing layout).

> **Note:**
>
> Local options for a layout may be changed only on the top level of the layout hierarchy, not from within a subcircuit.

When a new layout is created, options assume their global values, unless there is an installed default which overrides the global default. Changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing on the options in the **Layout Editor** tab) . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Choose options on the associated tabs, then click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

Setting Global Layout Options

The following options are set in the **Layout Editor** area of the Tools > Options > General Options window.

> **Note:**
>
> No project or layout design needs to be open to change global options. However, changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing on the options in the **Layout Editor** tab) . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

The following buttons control changes made to the global layout options:

- **Reset From Global Defaults** is used to change the settings of all the tabs to those of the current Global settings. The reset affects only the window contents.

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

- **Reset From Installed Defaults** is used to change the settings of all the tabs to those of the initial-install defaults. The reset affects only the window contents.
- **OK** saves the currents settings of all the tabs to the Global defaults registry. This action is not undoable and is not scriptable. A window appears to allow you to confirm the changes in the global options by clicking **Yes**.
- **Cancel** closes the window without affecting the Global default settings. (Clicking **No** on the confirmation window also closes all window without changing the settings.)

Both the global and local **Layout Options** windows have six tabs. The options in each area are much the same for both global and local settings. The differences are:

- **Save as Global Defaults** is present only for each of the local options tabs.
- **Always Show Layer Merging window** on the **Other** area is present only for global options.
- **Fit to Contents** on the **Display** area is present only for local options.
- The **Set based on page extent** button on the **Advanced** area is present only for local options.

> **Note:**
>
> Saving global defaults is not undoable. When global defaults are saved, HFSS immediately writes them to the registry , then uses these new default values for all subsequent applications. The only way to revert to previous global values is to once again modify the default values , then do another global save.

Layout Editor Options: Object Panel

Set the following options from the **Design Settings** window by navigating to **Tools**> **Options**> General Options. From the options tree, expand **Layout Editor** and select **Object**.

> **Note:**
>
> Changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing options from the **Layout Editor** tab) . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

Drawing

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

- **Draw connection points** — when selected, enables the display of pin symbols in the layout. Unselected by default.
- **Label vertices on selected polygons and lines** — when selected, enables the display of labels (Pt_0, Pt_1, etc.) on vertices of selected polygons and lines in the layout. Unselected by default.
- **Color by net** — color geometry by net rather than layer. Colors are assigned in the net list window.
- **Display package graphics** — displays the outlines of the 3D component packages in use.

Symbol

- **Auto scale** — use the check box to toggle whether to auto scale the axis.
- **Scale factor** — scale factor for symbol footprints placed in the layout. Default is 0.1.

Rectangle description style

- Specifies the method for describing rectangles. Default is to use **Two points**. Alternative method uses **Center /width /height**.

Measurement distances

- Controls whether measurement displays in layout include a **Label with units**, and also how many **Significant digits** are displayed.

Vias

- **Suppress non-functional internal pads** — when this option is enabled, padstack pads are affected in the following ways:
  - Top/Bottom pads always draw
  - Pads connected to traces on the same net appears
  - Pads intersecting fill material draws
  - Pads not connected to fill or to traces on the same net are suppressed; they are draw and are not modeled
- From the drop-down menu, select one of the following:
  - **Render antipad for center point intersection** — when this option is selected, antipads are only rendered if the via center falls within fill material from another net. A via placed outside of a fill region still generates a clearance, even if the antipad would otherwise extend into nearby fill regions.
  - **Antipad always enabled** — when this option is selected, antipads are always rendered for all vias, regardless of their placement or surrounding fill material. This ensures that a clearance is created in all cases where a via is present.

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

- **Render antipad for pad intersection** — when this option is selected, antipads are only rendered when the via's pad intersects with fill material from another net. If the via's pad does not make contact with another net's fill region, no antipad is generated.

Other

- **Auto place components in free space** — when selected, components placed in a schematic are positioned in Layout in the first available empty position. With regards to CPU time, the empty-position algorithm is relatively expensive. Consequently, if the physical layout of the components is irrelevant, placement of components is faster when this option is deactivated.
- **Auto position microwave components** — when selected, enables the automatic alignment of microwave components placed in the layout. Selected by default.

Layout Editor Options: Snapping Panel

When creating or placing a component initially in the **Layout Editor**, it is the cursor that snaps to the target. As you move and place objects, the shape of the cursor changes to indicate the target being snapped to. When multiple sources or targets are selected, the closest of these snap to each other as you create and move objects. **Sources** defines the part of the object you are placing that snap, and **Targets** defines what is snapped to. The options you may select under **Targets** and **Sources** are not mutually exclusive, and you may select none, some, or all them. The following options are set in the **Snapping** area under **Layout Editor** in the **Tools** > **Options** > General Options... window. These options control automatic snapping to objects, grid points and directions.

> **Note:**
>
> Changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing on the options in the **Layout Editor** tab) . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

Constraints

- **Snap To Grid** — targets that are not on the grid is ignored. If there are no usable targets in the snapping distance, the closest grid location is used.
- **Snap Across Hierarchy** — any object in the hierarchy becomes a target for snapping; this does not affect the sources.

Directional Constraint

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

- **None** — no directional drawing/snapping constraint
- **Orthogonal** — forces directional drawing/snapping to the orthogonal (x or y)
- **45 Degree** — forces directional drawing/snapping to 45 degrees

For more information see Directional Drawing.

Distance

Select the distance in pixels within which snapping occurs. Default is 20 pixels.

Targets

Select the types of targets to which objects can snap. By default, all targets are subject to snapping except **Any Edge Location** and **Edge Intersection**.

Sources

Select the types of source points that can be snapped. When **Object Center** is selected, it is the *center* of an object that snap to a target. For instance, with an arc, when **Object Center** is selected, the center of the arc snap to the nearest target point. By default, all sources are subject to snapping except **Any Edge Location**.

All snapping modes can be controlled via the **Layout Snaps** toolbar, which are active during an operation.

Layout Editor Options: Display Panel

The following options are set in the **Display** area under **Layout Editor** in **Tools** > **Options** > General Options. These options control the grid display, the drawing area, and the drawing mode.

> **Note:**
>
> Changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing on the options in the **Layout Editor** tab) . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

- **Major Divisions** – by default, the major grid divisions are spaced 10mm apart, and are a medium grey color. Reset the spacing by entering a value in the **Size** area. Select a new color on the palette that appears when you click in the color rectangle.
- **Minor Divisions** – by default, the major grid divisions are spaced 1mm apart, and are a light grey color. Reset the spacing by entering a value in the **Size** area. Select a new color on the palette that appears when you click in the color rectangle.

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

- **Background** – by default, the layout editor background is white. Select a new color on the palette that appears when you click in the color rectangle.
- **Show Grid Divisions** – toggles the grid display on and off. Default is on.
- **Drawing Extent** – specifies the size and extent of the layout. Defaults are Left, -100mm, Right, +100mm, Top, +100mm, Bottom, -100mm. Scroll bars indicate the extent of the full drawing area.
- **Drawing** – by default, the drawing mode for planar objects is to fill them with a crosshatched pattern. You can also select **Sketch Mode** (show outlines only) or **Solid Mode** (solid fill color)

Layout Editor Options: Selection Panel

These options control the colors and mode for selected objects.

> **Note:**
>
> Changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing from the options in the **Layout Editor** tab). To configure options for the **active** design, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes.

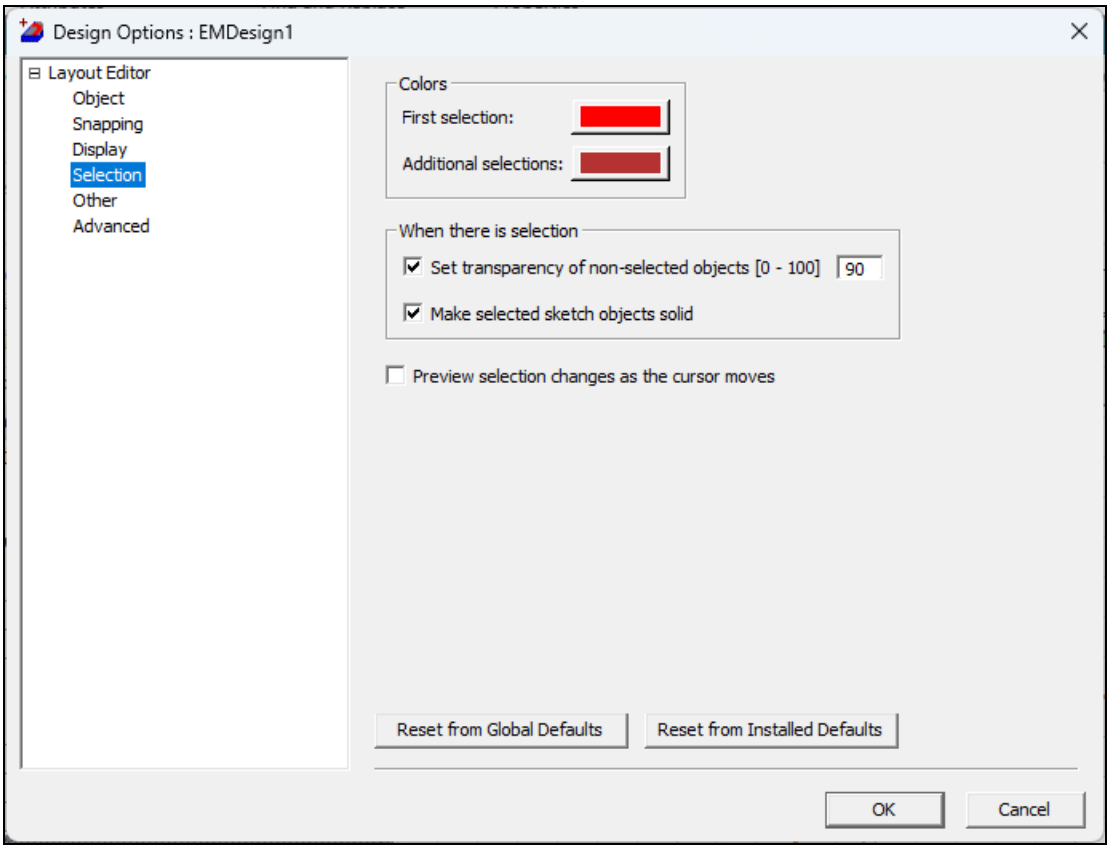
Follow these steps to reach the **Layout Editor Selection** options.

1. Navigate either of the following paths:

- To change new layout designs, navigate to **Tools** > **Options** > **General Options** > **Layout Editor**.
- To change an active design's settings, navigate to**Layout** > **Settings**.

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

2. From the **Layout Editor** tree, click **Selection**.



- **Colors** – by default, the first object selected (i.e., **First Selection**) is colored bright red, and **Additional Selections** turn dark red. Click the adjacent palette icons to select new colors.
- **When there is selection** area – specified objects can be set to transparent or solid to highlight them. Check both/either of the following boxes, as appropriate:
  - **Set transparency of non-selected objects** – automatically differentiates between the selected object and non-selected objects by making non-selected objects transparent. Enter a transparency percentage in the adjacent field (default is **90**).
  - **Make selected sketch objects solid** – makes selected objects solid, while non-selected objects remain outline only.
- **Preview Selection Changes as the Cursor Moves** — changes the color of an object to the **First Selection** color as the cursor moves across it.

Layout Editor Options: Other Panel

The following options are set in the **Other** area under **Layout Editor** in the **Tools** > **Options** > General Options window. These options control dragging, the Merge layers window, and

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

rotation. Not all these options appear in the local Layout options.

> **Note:**
>
> Changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing on the options in the **Layout Editor** tab) . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

- **Allow drag on first click** — when selected, holding down the mouse button on an object selects and attaches the object to the cursor. Drag the object to the selected location and release the mouse button. When unselected, first click to select the object, then click and hold down the mouse button to drag the object. Unselected is the default.
- **Show pin connectivity window on drag to matching component** — when selected, the **Pin Connectivity** window appears during a drag to a matching component.
- **Create ports when placing new component** — when selected, newly placed HFSS 3D Layout sub-designs, 3D Components, and HFSS dynamic links has ports. This is similar to adding ports on an existing component.
- **Always Show Layer Merging Window** — when selected, the **Merge Layers** window appears every time an object with layers is placed in the layout (or in a schematic). When unselected, the **Merge Layers** window appears only when automatic layer mapping cannot be performed. See the Merge Layers window. Unselected is the default.
- **Show select technology window on new design** — when inserting a new HFSS 3D Layout or Circuit Design, users are prompted to **Choose Technology**. This option suppresses the display of the Choose Technology window upon the insertion of a new design. For more information see Inserting an HFSS 3D Layout.
- **Specify default technology** — allows users to select a default technology on the pull down menu.
- **Naming Convention** — allows users to control the generation of port and pin group names when using the interactive commands.
- **Rotation increment** — during rotation using handles (**View** > **Rotate**), objects are rotated by multiples of this amount. The default rotation amount is 5 degrees. 2D rotations using **Draw** > **Rotate** always rotate 90 degrees counterclockwise.

Changing Naming Conventions

There are multiple ways to access the Naming Convention window:

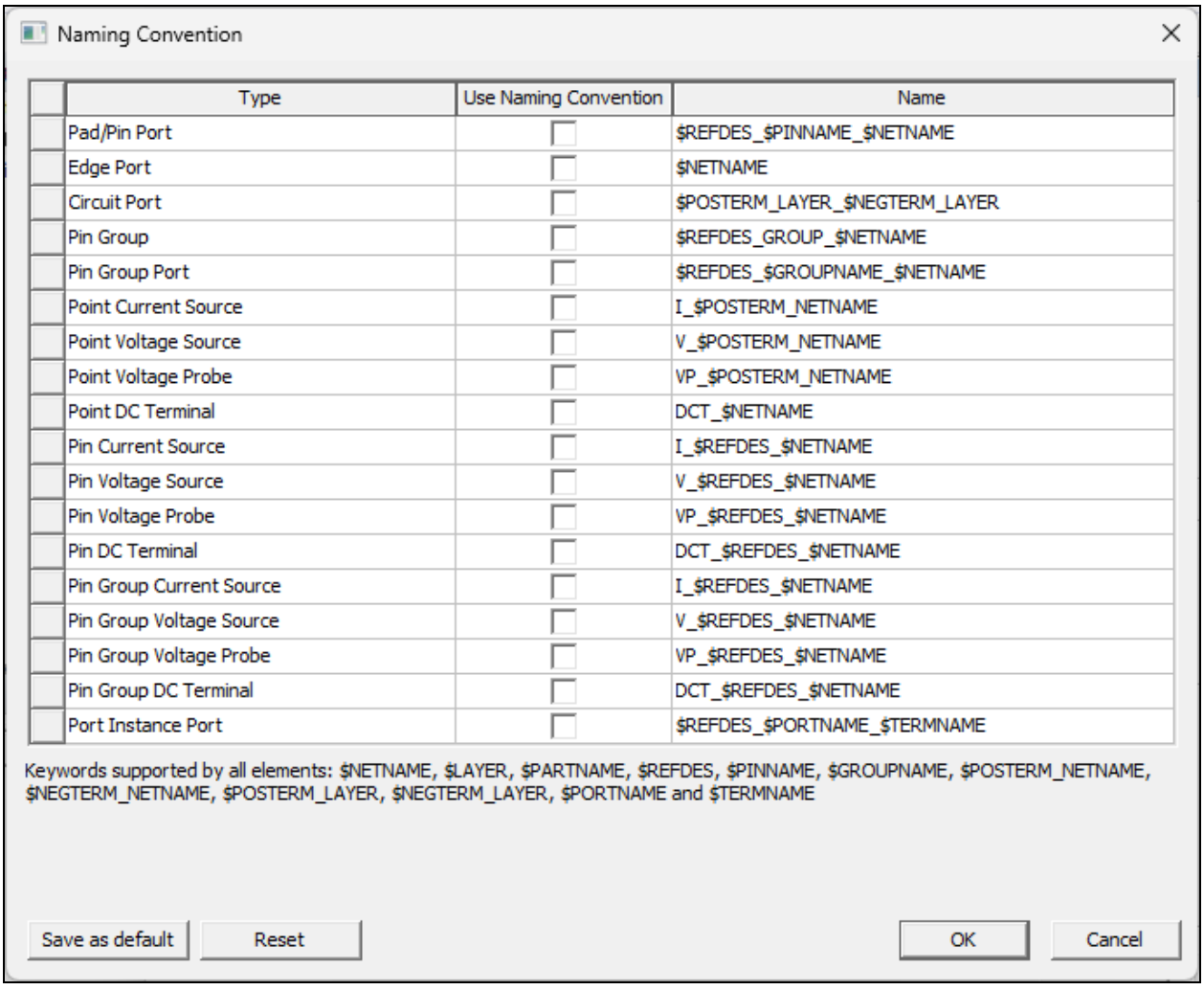
- From **Tools** > **Options** > **General Options** > **Layout Editor**.
- From the **General Options** icon on the **Desktop** tab.

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

- Through the SIwave Workflow Wizard, using either **Configure DC Drop Analysis** or **Configure PI Analysis**.

> **Note:**
>
> Only relevant naming convention types appear (e.g., if you access this window through **Configure PI Analysis**, only **Pin Group Port** naming conventions are editable). Access **Naming Conventions** through **General Options** for the greatest number of options.

The **Naming Convention** window lists the types of naming conventions that are editable:



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

1. Selecting a row displays its supported keywords underneath the list.
2. Double-click in the **Name** field to type your preferred naming configuration using the supported keywords.
3. Select a row's **Use Naming Convention** check box to apply the setting.
4. Click **Save as Default** to make your changes the default. Click **Reset** to restore default settings.

Example Naming Convention - Port Instance Port

HFSS 3D Layout has implemented naming convention support for multi-terminal ports exposed from MCAD (i.e., **Port Instance Port**). Assuming the convention is enabled, when a port is attached to an HFSS dynamic link or component (e.g., with Configuring Port-Instance Ports) the following default conventions are used.

Single port:

`$REFDES_$PORTNAME_$TERMNAME`

Multi-terminal ports:

`$REFDES_$PORTNAME_$TERMNAME:T1`

`$REFDES_$PORTNAME_$TERMNAME:T2`

Example — `S1_Port1_T1:T1, S1_Port1_T2:T2`

When a port name (e.g., `Port`*`N`*) is a prefix for a terminal name, the prefix is removed (e.g., if a terminal is named `Port1_T1, Port1` is removed, so that its port name will be `S1_Port1_T1` rather than `S1_Port1_Port1_T1`).

Users can also modify the assigned name for single and multi-terminal ports by adding additional keywords, such as the following: `$REFDES_$PORTNAME:T$TERMINDEX`

Example — `S1_Port1:T1, S1_Port1:T2`

Only the first part of the name is used for single terminal ports (i.e., the half of the name preceding the colon).

Example — `$REFDES_$PORTNAME`

If the specified convention is given a name without a colon (e.g., `$REFDES_$PORTNAME`), `:T1`, `:T2`, et cetera are automatically appended to the name to validate the multi-terminal port names.

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 Options: Advanced Panel

The following options are set in the **Advanced** area under **Layout Editor** in **Tools** > **Options** > General Options. These options control internal tolerance and resolution values.

> **Note:**
>
> Changes to global options only affect **new** layout designs (i.e., navigating to **Tools** > **Options** > **General Options** to open the global **Options** window, and choosing on the options in the **Layout Editor** tab) . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

- **Tolerance** – specifies the tolerance used internally by the layout editor. Distances are considered equal if they differ by no more than the tolerance value. Default is 1e-9 mm. Normally, this tolerance should not be changed.
- **Arc Drawing Resolution** – by default, arc resolution is dynamically adjusted depending on the zoom factor. You can instead specify a fixed resolution in the current drawing units (default units are mm).

Setting Local Layout Options

To change local layout options for a particular layout, on the **Layout** menu, select **Settings** or **Grid Settings** to open the **Tools** > **Options** > **General Options** window to the **Layout Editor** tab. Local options for a layout may be changed only on the top level of the layout hierarchy, not from within a subcircuit.

> **Note:**
>
> Changes to global options only affect **new** layout designs . To configure options for **existing** designs, navigate to the **Design Options** window (i.e., from **Layout**, select **Settings**). Then choose options on the associated tabs, and click **OK** to save changes. All changes made in the **Design Options** window affect **only** the active design).

When the window is accessed via **Grid Settings** it opens on the **Display** area. When the window is accessed via **Settings** it opens on the **Object** 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:**
>
> Saving global defaults is not undoable. When global defaults are saved, HFSS immediately writes them to the registry , then uses these new default values for all subsequent applications. The only way to revert to previous global values is to once again modify the default values , then do another global save.

The following buttons control changes made to the local layout options:

- **Reset From Global Defaults** is used to change the settings of all the tabs to those of the current Global settings. The reset affects only the window contents.
- **Reset From Installed Defaults** is used to change the settings of all the tabs to those of the initial-install defaults. The reset affects only the window contents.
- **Save As Global Defaults** saves the currents settings under all tabs to the Global defaults registry. This action is not undoable and is not scriptable. The save takes place immediately and does not close the window. Use of **Cancel** after **Save As Global Defaults** does not restore the registry.
- **OK** saves the settings of all the tabs to the current layout to be used locally, then closes the window. The action is undoable and is scriptable.
- **Cancel** closes the window without affecting the local settings of the layout.

Both the global and local **Layout Options** windows have six tabs. The options on each area are much the same for both global and local settings. The differences are:

- **Save as Global Defaults** is present only for each of the local options tabs.
- **Always Show Layer Merging window** on the **Other** area is present only for global options.
- **Fit to Contents** on the **Display** area is present only for local options.
- The **Set based on page extent** button on the **Advanced** area is present only for local options. This button calculates a distance tolerance based on the page extent.

Edit Menu

When a **Layout Editor** design is active, click the **Edit** menu. There are multiple varieties of this menu depending on the type of design being edited (Circuit, HFSS 3D Layout, etc). The following descriptions are a superset of the commands that may appear on the **Edit** menu, though they may appear in different order on the menu than the order in which they are presented here. For more information, search the Help or use the Help Index to find information on any of the commands that appear on the **Edit** menu for the Layout Editor.

Undo<*last edit operation*> — undo the last edit operation.

Redo<*last edit operation*> — redo the last edit operation.

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

Cut — delete the selected object, and retain a copy for pasting into a layout in the same application.

Copy — create a local copy for pasting into a layout in the same application.

Paste — put the object on the last cut or copy into the layout.

Delete — delete the selected object without retaining a copy.

Rename — click to rename the selected object in the layout.

Select All — select all objects in the layout.

Unselect All — unselect all objects in the layout.

Cycle Selection — cycle selection of two or more objects that overlap. Each time you click Cycle Selection, a different one of the overlapping objects is selected.

Invert Selection — unselect all objects that are currently selected, and select all the other objects.

Select by Polygon Region — selects entities intersecting a polygonal region.

Copy Image — create a global copy of the selected objects on the clipboard for pasting into a different application.

Copy to HFSS 3D Layout — copy the selected objects to the clipboard and paste into an HFSS 3D Layout Design.

Paste in Place — paste object to same location, without any X,Y offset/displacement.

Select — sets cursor mode to selection of objects.

Select Handles — sets cursor mode to selection of object handles.

Select Edges — sets cursor mode to selection of object edges.

Properties — opens the **Properties** window to view and edit the properties of the selected object.

Activate — causes the activation of one or more selected components.

Deactivate (Open) — temporarily converts the component into an open circuit. Activate restores a deactivated component to the circuit.

Deactivate (Short) — temporarily converts the component into a short circuit. Activate restores a deactivated component to the circuit.

Pop Up — switches the active coordinate system (CS) to the next higher level. Relative CSs can be created within other relative CSs to form multiple relative levels.

Push Down — switches the active CS to the next lower level.

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

Port — opens a drop-down menu that contains commands for operating on Ports.

Padstack — opens the Edit Padstack Definition window.

Footprint — opens the Footprint Editor.

View — opens the **Layout Editor** View drop-down menu.

Draw — opens the **Layout Editor** Draw drop-down menu.

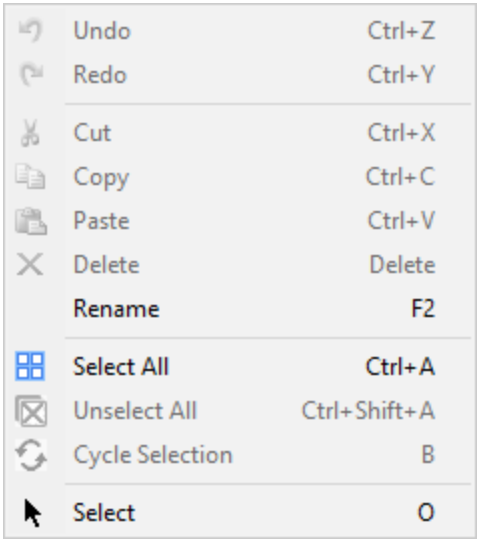
Edit — opens the **Layout Editor** Edit drop-down menu.

Copy Image — create a global copy of the selected objects on the clipboard for pasting into a different application.

EM Design Properties — set EM design properties via the **EM Design Properties** drop-down menu.

IC Mode Commands

The following options appear when the IC mode environment is active.



Undo<*last edit operation*> — undo the last edit operation.

Redo<*last edit operation*> — redo the last edit operation.

Cut — delete the selected object, and retain a copy for pasting into a layout in the same application.

Copy — create a local copy for pasting into a layout in the same application.

Paste — put the object on the last cut or copy into the layout.

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

Delete — delete the selected object without retaining a copy.

Rename — click to rename the selected object in the layout.

Select All — select all objects in the layout.

Unselect All — unselect all objects in the layout.

Cycle Selection — cycle selection of two or more objects that overlap. Each time you click Cycle Selection, a different one of the overlapping objects is selected.

Select — sets cursor mode to selection of objects.

View Menu

When a **Layout Editor** design is active, clicking the **View** button on the menu to open the view drop-down menu for the Layout editor. There are multiple varieties of this menu depending on the type of design being edited (Circuit, HFSS 3D Layout, etc). The following descriptions are a superset of the commands that may appear on the **View** menu, though they may appear in different order on the menu than the order in which they are presented here. For more information, search the Help or use the Help Index to find information on any of the commands that appear on the **View** menu for the Layout Editor.

Display — control the display by selecting the check box of the following: **Status Bar**, **Project Manager** window, **Message Manager**, **Project Manager**, **Properties Window**, **Progress Window**, **Component Manager**, **Layout Window.**

Variables — view the project and design variables with their values in a grid view.

Display Mode — select on the following rendering options:

- **Default** — honors transparency settings configured in stackup
- **Sketch** — very similar to wireframe
- **Solid** — honors the wireframe settings per layer

Rotate — **click+drag** in any direction to rotate the view; the same double-click options hold true for the **Alt** +drag editor display options. For more information see Layout Editor Display.

Zoom — change the zoom display by selecting one of the following: **Zoom In, Zoom Out, Zoom Area, Zoom Previous.**

Fit or Pan — fit or pan by selecting: **Fit Drawing, Fit Selection, Pan.**

ResetOrientation — reset the rotation to a simple top-down view: positive Y up, positive X to the right.

Faster Transformations — deactivate the rendering of padstacks (vias and pins) during panning, zooming, and screen rotations; this increases the responsiveness of these interactive operations.

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

Stretch — selecting **Stretch Z** opens a window that allows you to apply a scaling factor. Dragging the slider applies a Z scaling factor to the current view. If **Dynamic** is selected, the scaling happens in real time; otherwise, it is applied when **OK** is selected.

IC Mode Commands

The following options appear when the IC mode environment is active.

Display — control the display by selecting the check box of the following: **Status Bar**, **Project Manager**, **Message Manager**, **Properties**, **ACT Extensions**, **Component Libraries**, **Variables**, **Layers**, **Nets**, **Components**, and **Cells**.

Docking Window Layouts — choose either to open the **Default** windows, **Save Current Layout** (i.e., save the current **Display** configuration), or **Remove Saved Layouts**.

Display Mode — select on the following rendering options:

- **Default** — honors transparency settings configured in stackup
- **Sketch** — very similar to wireframe
- **Solid** — honors the wireframe settings per layer

Rotate — **click+drag** in any direction to rotate the view; the same double-click options hold true for the **Alt** +drag editor display options. For more information see Layout Editor Display.

Zoom — change the zoom display by selecting one of the following: **Zoom**, **Zoom Area**, **Zoom In**, **Zoom Out**, or **Zoom Previous**.

Fit or Pan — fit or pan by selecting: **Fit All**, **Fit Selection**, or **Pan**.

Stretch Z — opens a window that allows you to apply a scaling factor. Dragging the slider applies a Z scaling factor to the current view. If **Dynamic** is selected, the scaling happens in real time; otherwise, it is applied when **OK** is selected.

Reset Orientation — resets the rotation to a simple top-down view: positive Y up, positive X to the right.

Clip Plane — see Clip Planes.

Draw Menu

When a **Layout Editor** design is active, click the **Draw** menu. There are multiple varieties of this menu depending on the type of design being edited (Circuit, HFSS 3D Layout, etc).

The following descriptions are a superset of the commands that may appear on the **Draw** menu, and depending upon the context of the Ansys Electronics Desktop (AEDT)™ electronics systems design platform's state, they may appear in different order and number on the menu than the order in which they are presented here.

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, search the Help or use the Help Index to find information on any of the commands that appear on the **Draw** menu for the Layout Editor.

Primitive — opens a drop-down menu that allows you to create a primitive object including bondwires in the Layout Editor.

Void — opens a drop-down menu that allows you to create a void in the Drawing Region.

Merge Polygons — opens a drop-down menu that allows you to operates on two or more objects that overlap.

Port — opens a drop-down menu that allows you to place a port in the Drawing Region.

Boundary — opens a drop-down menu that allows you to place or remove a boundary excitation on the Drawing Region.

Pin — place a Pin in the Drawing Region.

Create/Manage Pin Groups — opens a window that allows you to create pin groups for various components.

Via — place a Via in the Drawing Region.

Via Groups — allows primitives to be grouped together and represented by a single primitive.

3D Structure — opens a drop-down menu that allows you to create and perform operations on a 3D structure in the Drawing Region.

Coordinate System — opens a drop-down menu that allows you to create and perform operations on a coordinate system in the Drawing Region.

Component — opens a drop-down menu that allows you to create and remove a component.

Rotate — rotate a selected object in the Drawing Region.

Flip Vertical — vertically flip a selected object in the Drawing Region.

Flip Horizontal — horizontally flip a selected object in the Drawing Region.

Reverse Line — reverses the direction of selected Bond Wires, Lines and Arcs.

Align — align selected objects in the Drawing Region.

Position Relative — when a relative coordinate system (CS) and an object independent of that coordinate system (CS) are selected, this command positions the CS relative to that object. The 0,0 origin of the relative CS is placed alongside the positioning object. There are three types of objects select and position the CS relative to: an edge port, a pin, or an edge.

Clear Relative Position — when a relative CS and an object are selected, this command positions the CS unrelative 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.

Duplicate — duplicate a selected object in the Drawing Region via the Duplicate drop-down menu.

Expand — expand a selected object in the Drawing Region.

Split Polygon Region — split primitives at a polygon boundary; primitives are split and not clipped.

Geometry Healing — heal and repair selected objects in the Drawing Region.

Sanitize Layout - fixes alignment problems and other complexities in power and ground nets that may slow down simulation

Convert — opens a drop-down menu that allows you to create vias, pins, and padstack definitions from primitives, convert traces to planes, and convert planes to traces.

Stitch Lines — stitch together connected/crossing lines into polygons and lines.

Connection — connect selected objects to the Net.

Disconnect from Net — disconnect selected objects on the Net.

Route — Deprecated. Opens a drop-down menu that allows you to establish traces between different circuit elements and set Automatic Trace Drawing Options.

Align MW Ports — align selected MW ports in the Drawing region.

HFSS 3D Layout Properties — opens a drop-down menu that allows you to define HFSS 3D Layout Properties.

IC Mode Commands

The following options appear when the IC mode environment is active.

Primitive — opens a drop-down menu that allows you to create a primitive object in the **Layout Editor**.

Void — opens a drop-down menu that allows you to create a void in the **Layout Editor**.

Merge Polygons — opens a drop-down menu that allows you to operates on two or more objects that overlap.

Port — opens a drop-down menu that allows you to place a port in the **Layout Editor**.

Boundary — opens a drop-down menu that allows you to place or remove a boundary excitation on the **Layout Editor**.

Create/Manage Pin Groups — opens a window that allows you to create pin groups for various components.

Via Groups — allows primitives to be grouped together and represented by a single primitive.

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

Component — opens a drop-down menu that allows you to create and remove a component.

Duplicate — duplicate a selected object in the **Layout Editor** via the **Duplicate** drop-down menu.

Expand — expand a selected object in the **Layout Editor**.

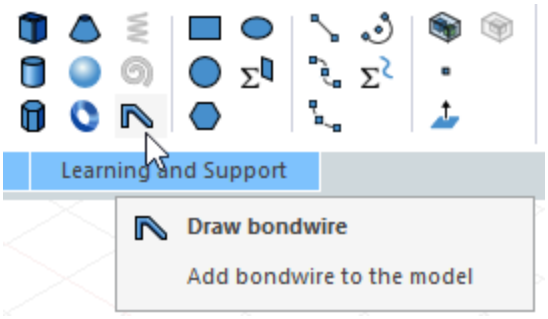
Convert — opens a drop-down menu that allows you to create vias, pins, and padstack definitions from primitives, convert traces to planes, and convert planes to traces.

HFSS 3D Layout Properties — opens a drop-down menu that allows you to **Add Modeling** or **Remove Modeling** from a design component.

Drawing a Bondwire in HFSS 3D Layout

A bondwire is a thin metal wire that connects a metal signal trace with a chip.

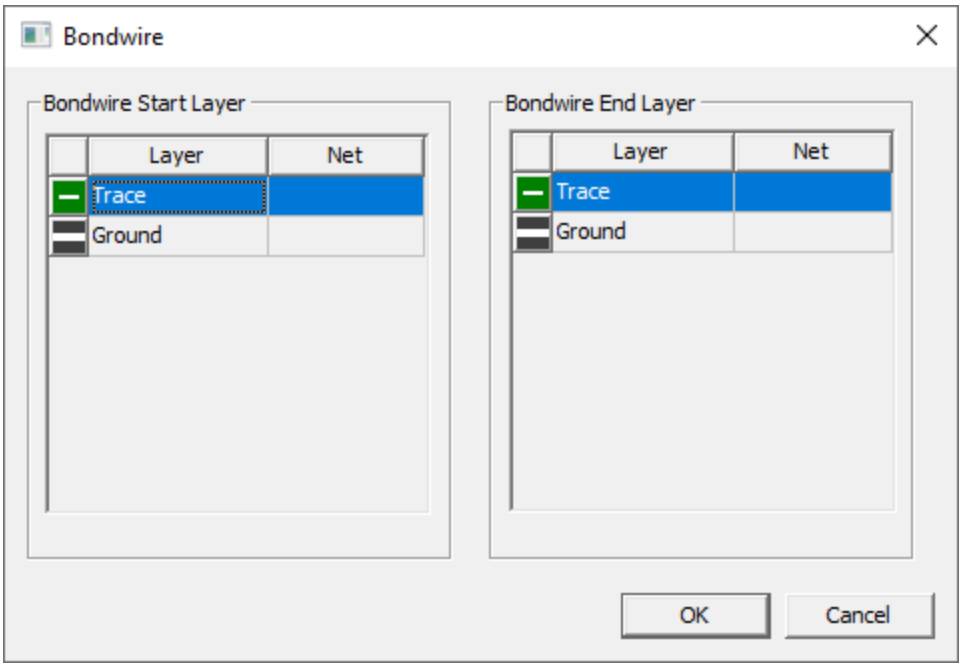
1. From the menu bar, click **Draw** > **Primitive** > **Bondwire** or, on the **Draw** ribbon tab, click the **Draw bondwire** icon:



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.

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 lead point by clicking the point or typing the coordinates in the text boxes in the status bar. The **Bondwires** dialog box opens.

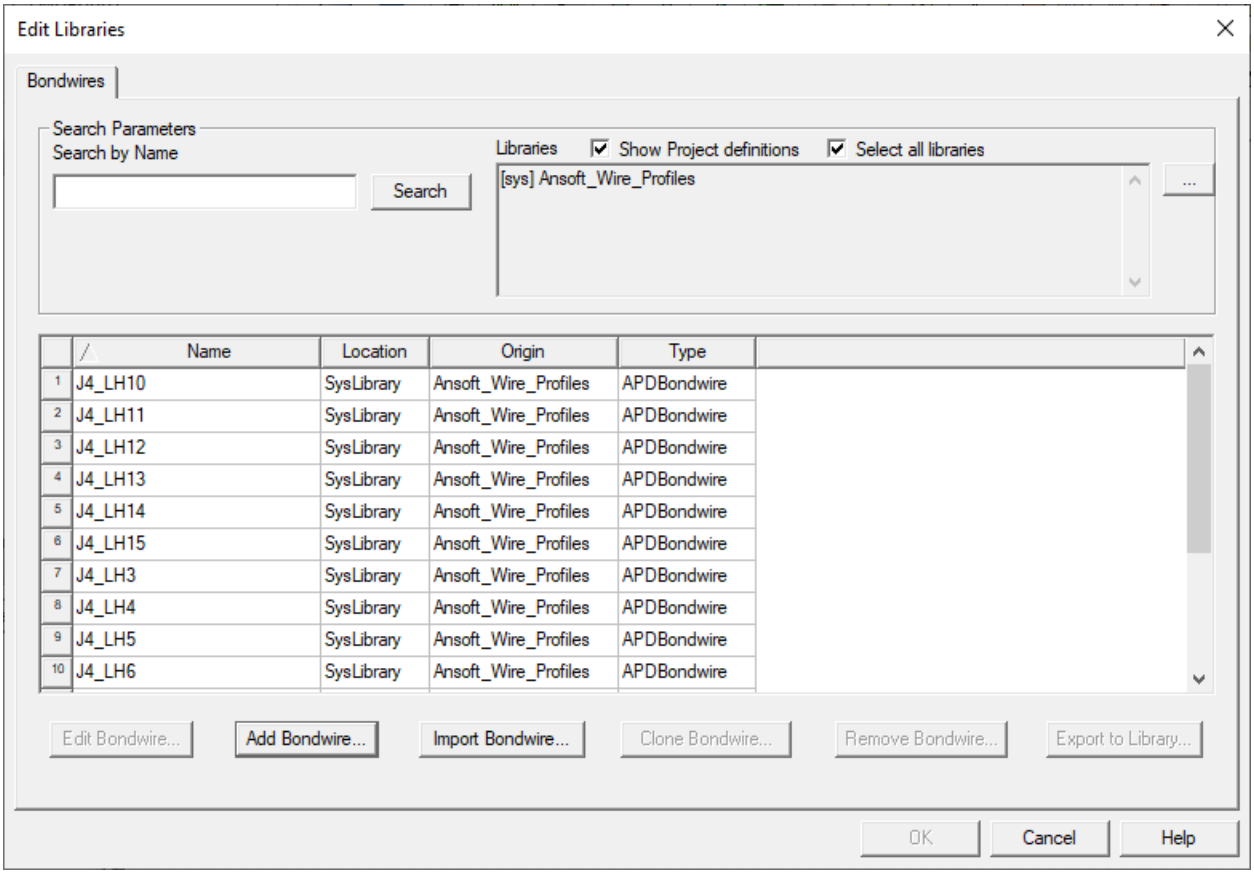


4. Select which layers you want the bondwire to start and stop on.
5. Click **OK**. The bondwire appears in the Layout editor.
6. To select a new profile definition, click in the **Type** row of the **Properties** window and select a modeling standard shape. The choices are **APDBondwire**, **JEDEC4Bondwire**, and **JEDEDC5Bondwire**.
7. To change the underlying profile definition, click in the **Profile** row of the **Properties** window and click **Edit**. The **Bondwires** tab of the **Edit Libraries** window opens where you can add, edit, import, export, and delete bondwire profile definitions.

Bondwire Definitions

To view a list of bondwire definitions, on the menu bar click **Tools** > **Edit libraries** > **Bondwires** to open the **Bondwires** tab of the **Edit Libraries** window. From this window, add, edit, import, clone, remove, or export a bondwire definition.

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

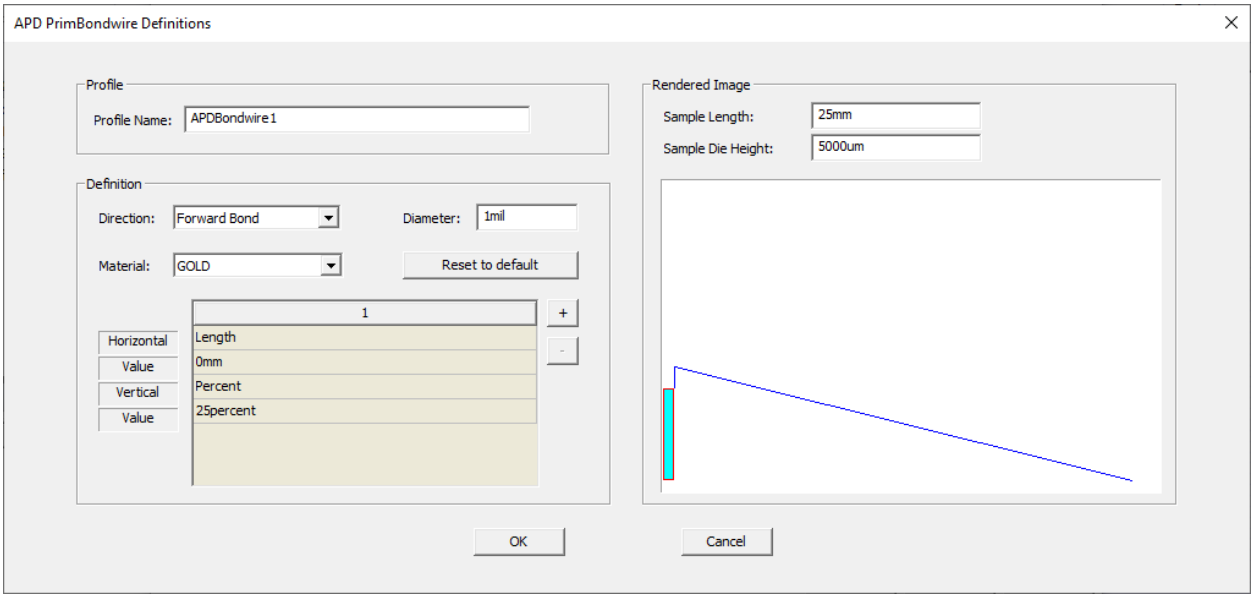


Add a Bondwire Definition

1. Click **Add Bondwire** to open the **New Bondwire Definition** window.
2. Optionally, enter a new name for the definition to replace the default.
3. From the **Type** field, select a type of bondwire profile definition.
4. Click **OK** to open the **Bondwire Definitions** window with default values.

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

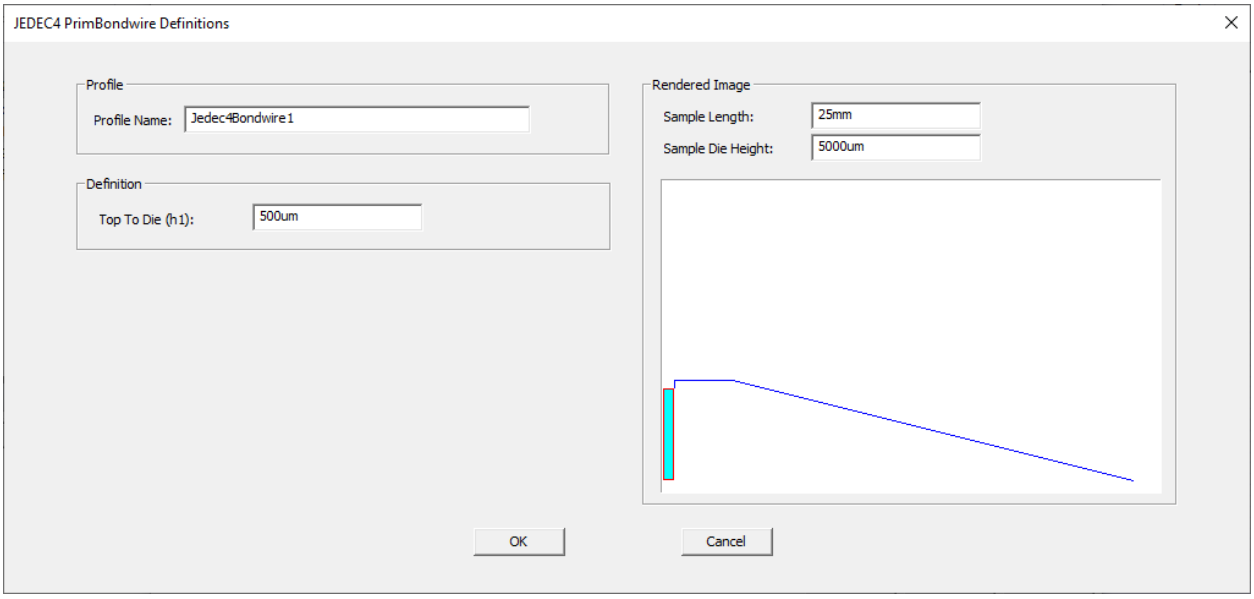
5. For **APD PrimBondwire Definitions**:



a. Select a **Direction**.
b. Enter a **Diameter** with its units.
c. Choose a **Material**.
d. Select **Edit** to open the **Materials** tab of the **Select Definition** window.
e. To add a segment, click **+** and adjust the segment's settings. All the changes are reflected in the bondwire graphic.
f. To remove a segment, the number of the segment and click **–**.
g. Click **OK**.

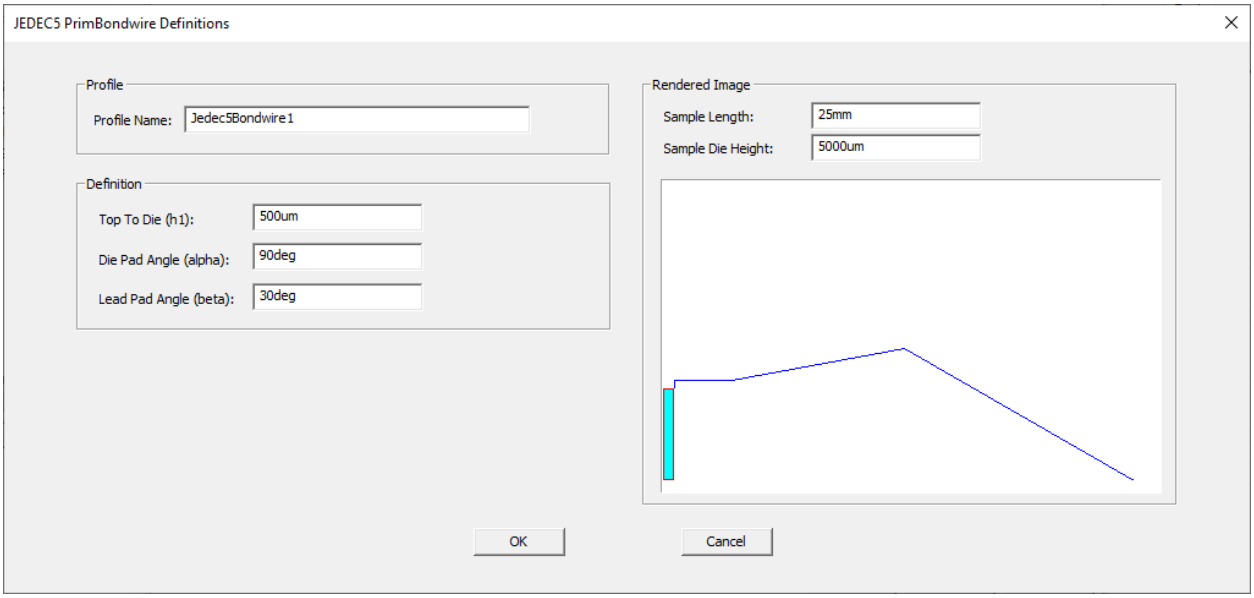
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. For **JEDEC4 Prim Bondwire Definitions**:



   a. Enter the height from **Top to Die** including its unit of length. This is the height between the bond pad and the top of the loop.
   b. Click **OK**.

7. For **JEDEC5 PrimBondwire Definitions**:



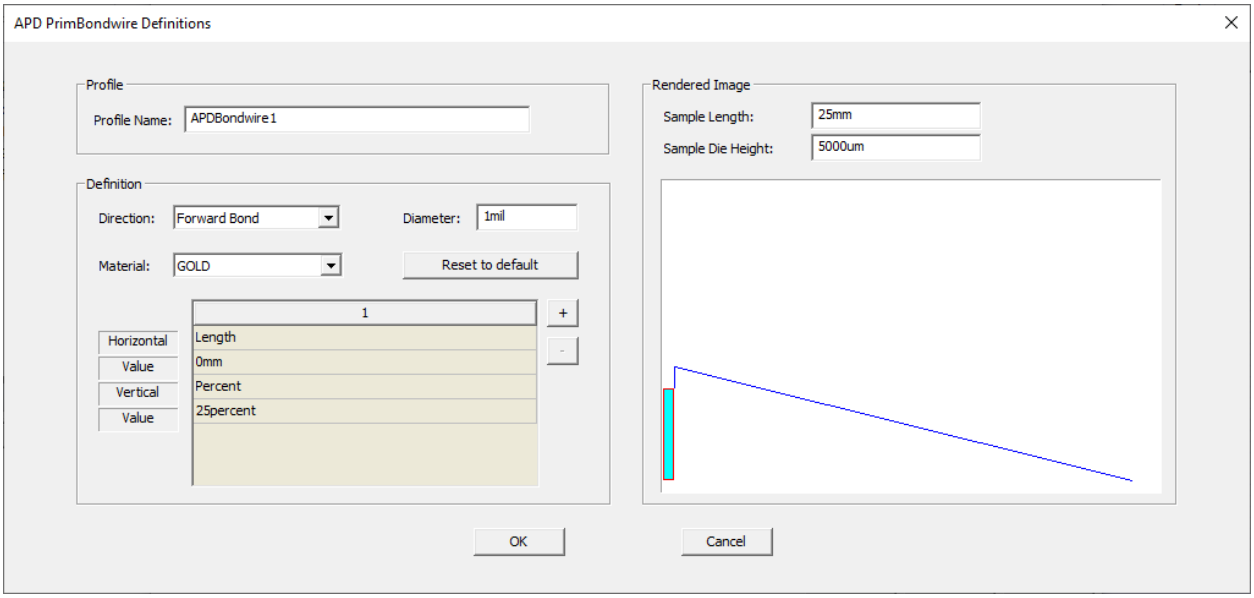
   a. Enter the height from **Top to Die** including its unit of length. This is the height between the bond pad and the top of the loop.

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

b. Enter the **Die Pad Angle** which is the angle between the horizontal plane and the wire at the bond pad point. Include the unit **deg**.
c. Enter the Lead Pad Angle which is the angle between the horizontal plane and the wire at the lead point. Include the unit **deg**.
d. Click **OK**.

Editing a Bondwire Definition

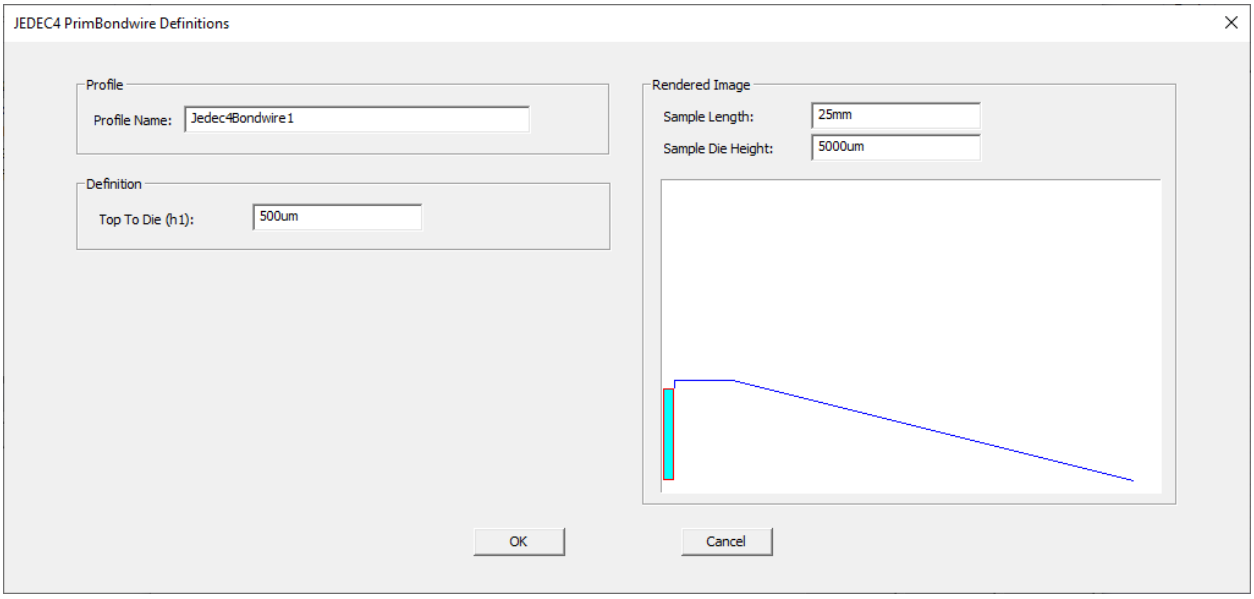
1. Select the profile definition you want to edit and click **Edit Bondwire** to open th **Bondwire Definitions** window.
2. For **APD PrimBondwire Definitions**:



a. Choose a **Direction**.
b. Enter a **Diameter** with its units.
c. Choose a **Material**.
d. Select **Edit** to open the **Materials** tab of the **Select Definition** window.
e. To add a segment, click **+** and adjust the segment's settings. All the changes are reflected in the bondwire graphic .
f. To remove a segment, the number of the segment and click **–**.
g. Click **OK**.

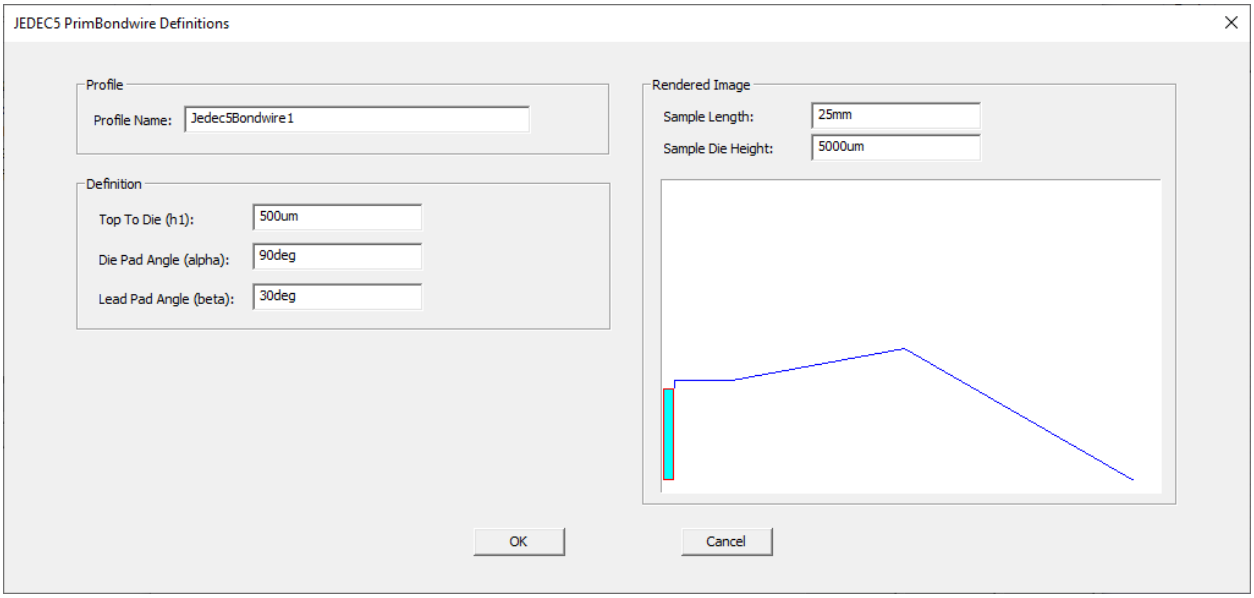
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. For **JEDEC4 Prim Bondwire Definitions**:



   a. Enter the height from **Top to Die** including its unit of length. This is the height between the bond pad and the top of the loop.
   b. Click **OK**.

4. For **JEDEC5 PrimBondwire Definitions**:



   a. Enter the height from **Top to Die** including its unit of length. This is the height between the bond pad and the top of the loop.

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

b. Enter the **Die Pad Angle** which is the angle between the horizontal plane and the wire at the bond pad point. Include the unit **deg**.
c. Enter the Lead Pad Angle which is the angle between the horizontal plane and the wire at the lead point. Include the unit **deg**.
d. Click **OK**.

Importing Cadence Bondwire Profiles

1. Click **Import Bondwire** to open an explorer window.
2. Navigate to and select a Cadence XML wirebond type definition files
3. Click **OK**.

Cloning a Bondwire Definition

1. From the grid, select a bondwire definition to clone.
2. Click **Clone Bondwire**. The **Bondwire Definitions** window opens with values identical to the selected definition.
3. Click **OK**.

Removing a Bondwire Definition

1. From the grid, select a bondwire definition to delete. *Only project definitions can be deleted.*
2. Click **Remove Bondwire**.
3. Confirm the removal. The definition is deleted on the grid.

Export Bondwire Definitions to File

1. From the grid, select a bondwire definition to export. *Only project definitions can be exported.*
2. Click **Export To Library**.
3. From the **Export to Library** window, navigate to where you want the file to go and click **Save**.
4. Confirm the export.

Drawing a Void in the Layout Editor

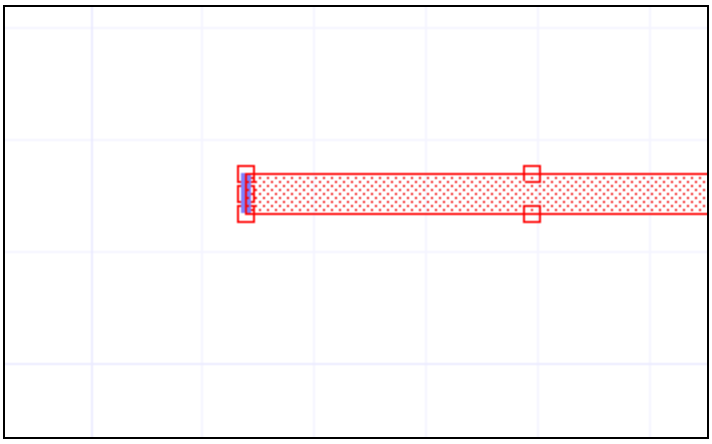
Follow these steps to create a void, or remove material from an object in the shape of a circle, rectangle, polygon, or line.

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

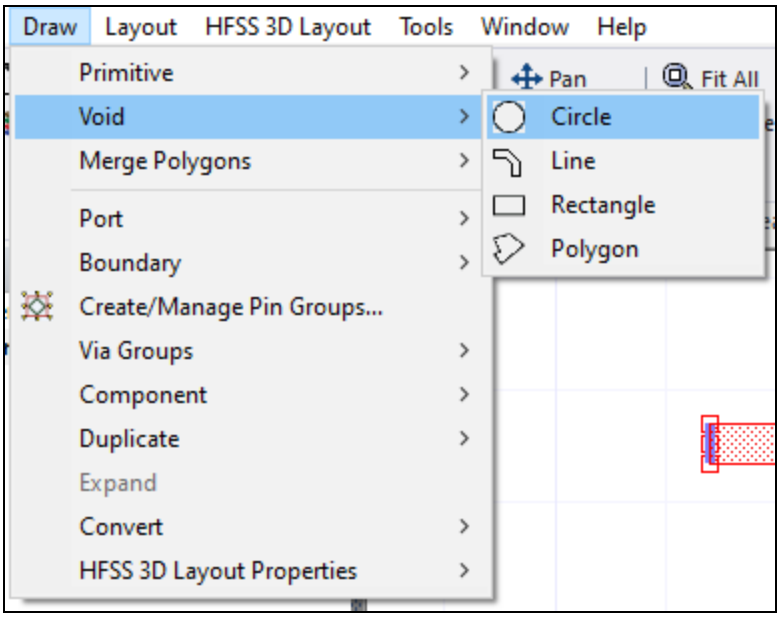
Note:

One object can have multiple voids but each void can only be associated with one parent object (e.g., to create two concentric rings, draw the outer circle, then add a circular void within it).

1. Select a parent object.



   a. Navigate to **Draw**, then select **Void** and one of the following:

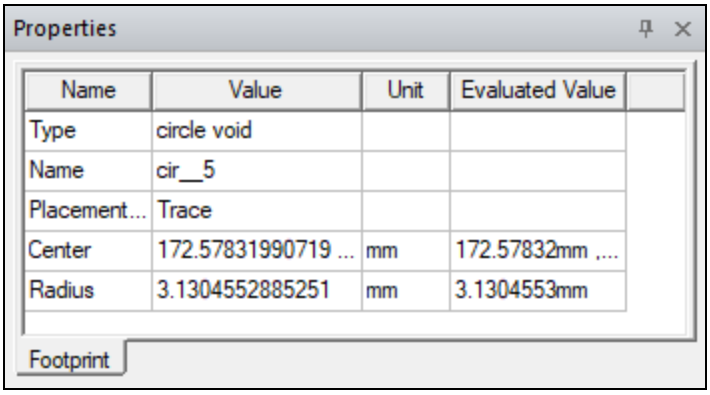


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Circle**. Then follow the procedure for drawing a circle.
- **Rectangle**. Then follow the procedure for drawing a rectangle.
- **Polygon**. Then follow the procedure for drawing a polygon.

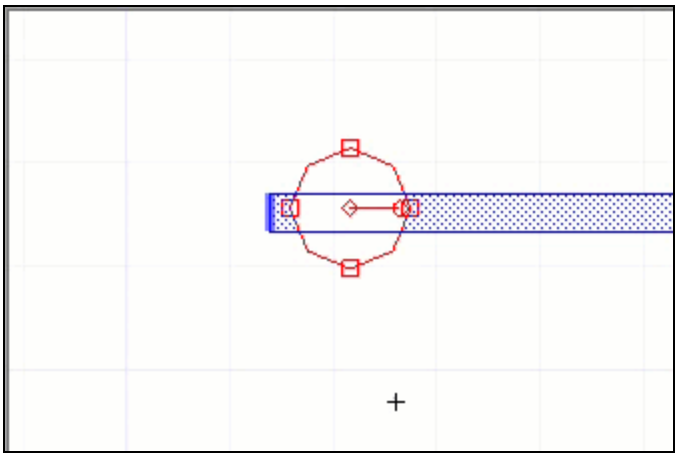
> **Note:**
>
> To **Remove a Line**, navigate to **Draw** menu, then select **Void** > **Line**. Follow the procedure for drawing a polygon. A void cannot be created in the shape of a line on an existing line object. Material is removed from the object in the shape and dimensions of the void.

2. Repeat steps **1-2** to create additional voids in an object.
3. View or edit a void's properties in the **Property Control** window or use the mouse to drag the void's handles.



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+drag** the void's handles to change its shape.



> **Note:**
>
> 3D vias may contain voids. Voids in 3D vias can be placed entirely or partially in the outer contour. No attempt is made to geometrically align voids in the upper polygons with those in the lower. The order in which voids on different layers are created is used to control their alignment.

Merge Polygons

Merge Polygons operates on two or more objects that overlap. Select the objects you want to merge, then click the **Layout** >**Merge Polygons** drop-down menu to select the operation:

- **Union** joins the objects into one object. See Uniting Objects.
- **Intersection** creates a new object from the intersection of the objects. See Creating Objects from Intersections.
- **Subtract** takes the object that was selected first as the positive object, then subtracts from it any overlapping areas on the other selected objects. See Subtracting Objects.

Union and **Intersection** operations support using selected objects on multiple layers, and each layer object is operated on separately. However, **Subtraction** is only allowed if the selected objects are on the same layer. Selected objects may be negative objects.

- If only negative objects are selected, the merge operation is performed the same as with positive objects, but the resulting objects remain negative objects.
- If both negative objects and positive objects are selected:
  - Complete the union/intersection/subtraction operation on the positive objects
  - Always union the negative objects

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

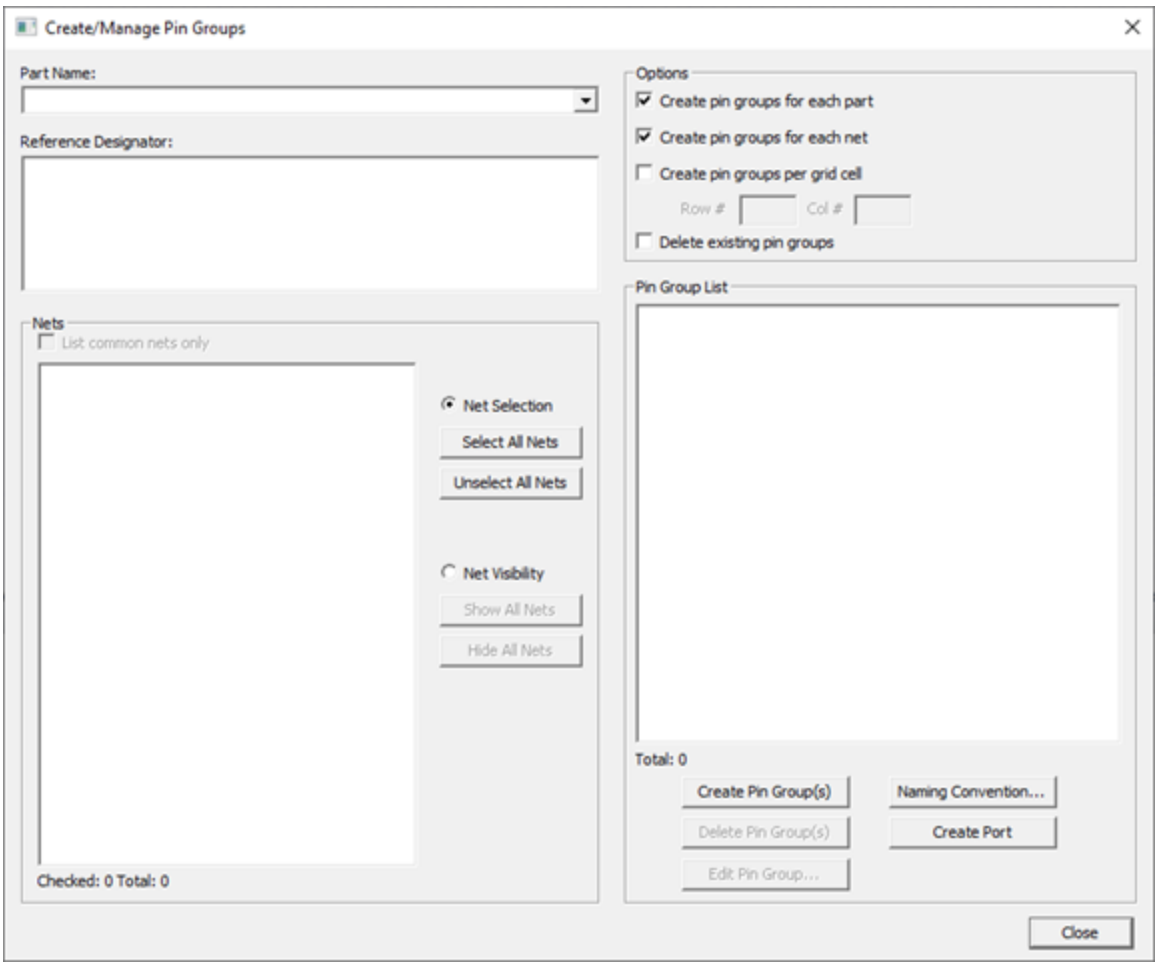
- Subtract the negative object result on the positive object result

Pin Groups in 3D Layout

Pin groups are groups of pins created for various components. Each pin group is treated as if it is electrically connected during analysis. You can connect circuit elements (e.g., ports, sources, RLCs) to any generated pin groups.

Pin groups are listed in the Components window. You can select a pin group or a pin in a group by clicking on the entity in the **Components** window.

Most of a pin group's configuration is in the **Create/Manage Pin Groups** window.

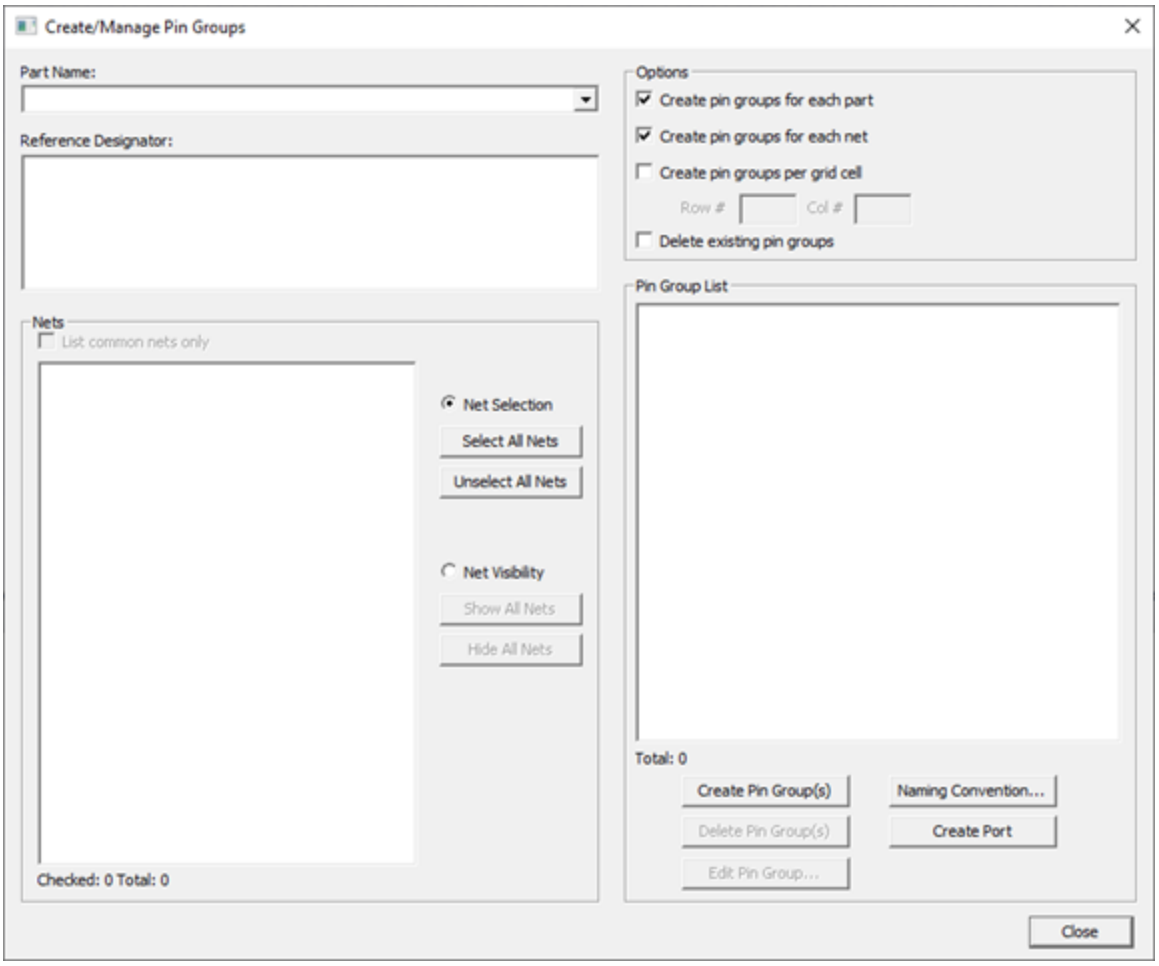


Creating a Pin Group

Pin groups can be created from pins that are both in components and assigned to a net. If a pin is not assigned to a net or a member of a component, it cannot join a group.

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

1. Create any appropriate components or nets.
2. Click **Draw** > **Create/Manage Pin Groups** to open the **Create/Manage Pin Groups** window.



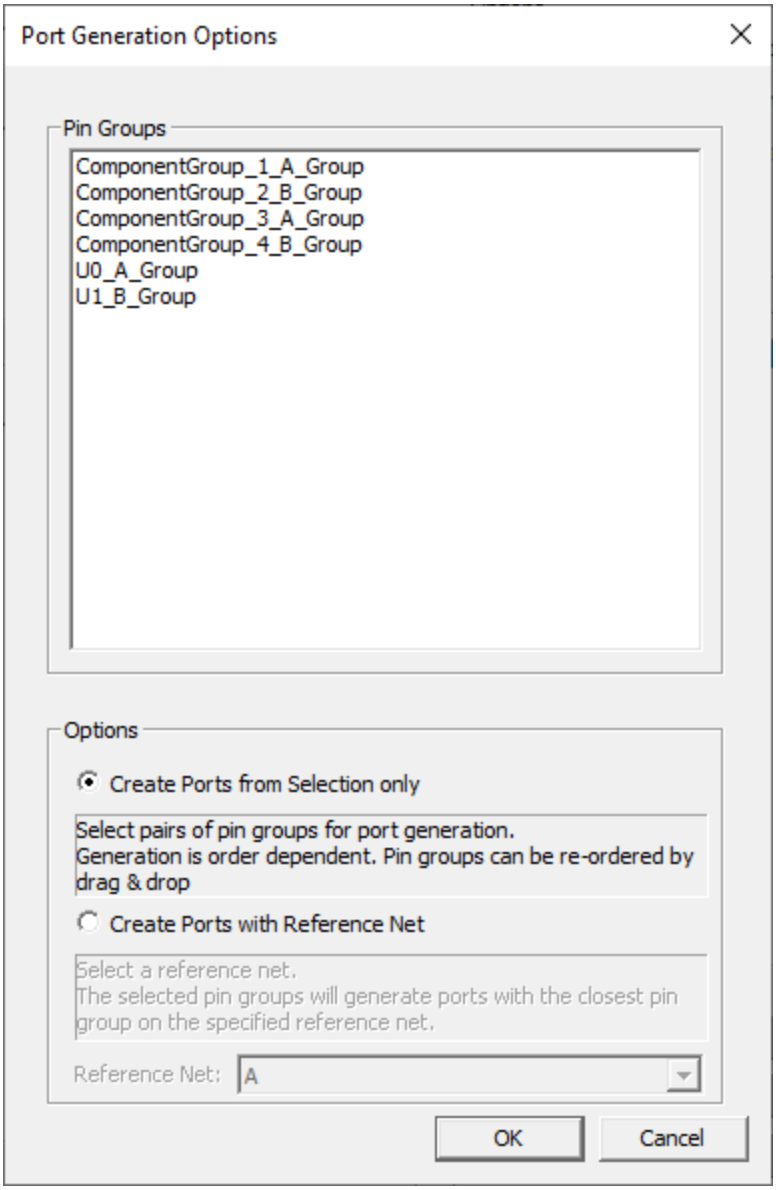
3. Make a selection on the **Part Name** drop-down menu. This is the component name. Select **Don't know** to display existing pin groups in the design.
   Nets that do not "overlap" with a component do not appear in this window.
4. Select the **Reference Designator** (s).
5. Select the **Net Selection** or **Net Visibility** options:
   - To select all nets that overlap with the component, click **Net Selection**, then click **Select All Nets**. The pins associated with the selected nets are shaded.
   - To show all the nets in the drawing area, click **Net Visibility**, then click **Show All Nets**.
6. To show only the common pins when multiple reference designators are selected, click **List common nets only**.

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 a selected **Part Name** contains multiple **Reference Designators**, select any or all them and click **Create pin groups for each part** to have identical pin groups created on each Reference Designator.
8. If multiple nets are selected, click **Create pin groups for each net** to generate multiple pin groups; one for each of the nets that are selected.
9. To specify the grid locations to create pin groups, select **Create pin groups per grid cell** and enter the **Row #** and **Col #**. The **Drawing Area** is updated.
10. To create pin groups from a reference designator:
    a. Select the nets you want to include in the pin group on the **Nets List**. All pins associated with the net(s) are included. When nets are selected on the net list box, the associated pins are marked with black mesh boxes.
    b. Specify if you want to use the naming conventions. Click **Naming Convention** to open the **Naming Convention** window. Select if you want to **Use the Naming Convention** for the particular circuit element. Click **OK** to close this window.
    c. Click **Create Pin Group**.
11. To create a pin groups of pins from multiple reference designators:
    a. Select the pins you want to include in the pin group on the **Drawing Area**.
    b. Clear the **Create pins groups for each part** and **Create pins groups for each net** to create a pin group.
    c. Click **Create Pin Group**.

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

12. To create ports, click **Create Port** to open the **Port Generation Options** window.



a. Drag the pin groups in the **Pin Group** list into the order that you want them to pair up. The first group pairs with the second, the third with the fourth, etc.
b. To create ports between selected pin groups, select an even number of groups in the list and click **Create Ports from Selection only**.
c. To create a port from a net, select one or more pin groups, click **Create Ports with Reference Net**, and select a net to reference. If you select multiple pin groups, each selection creates a port with that pin group and the closest pin group 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.

selected net.

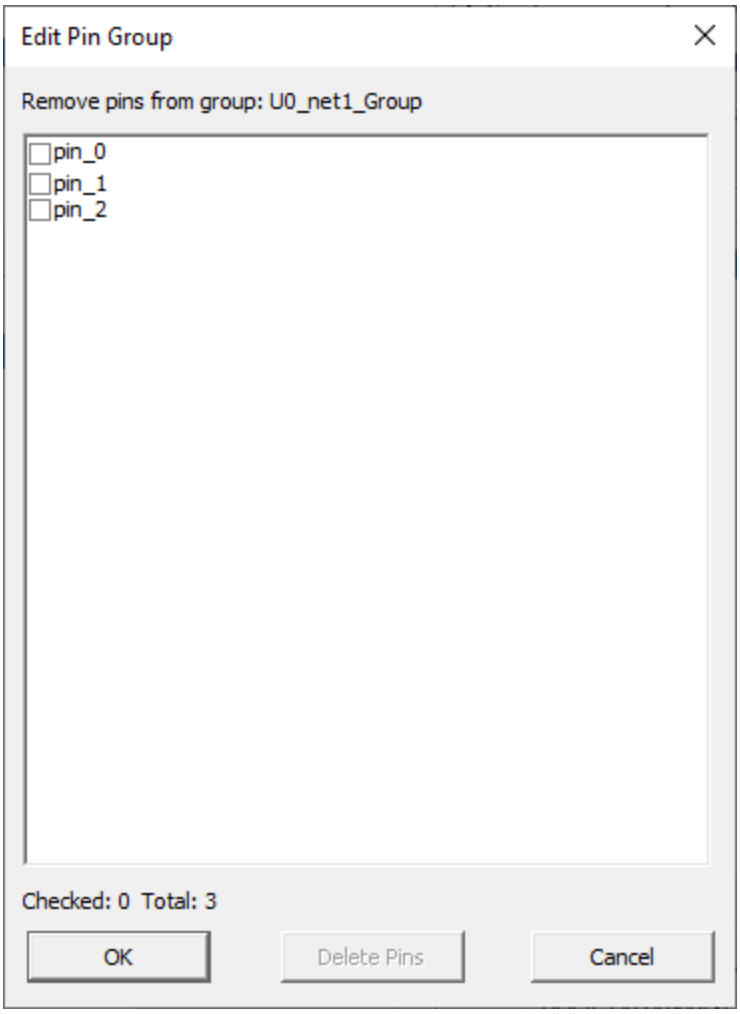
d. Click **OK**.

13. Click **Close**.

Editing a Pin Group

The only changes that can be made to a pin group is to remove member pins. Also rename a pin group in the **Components** window.

1. Click **Draw** > **Create/Manage Pin Groups** to open the **Create/Manage Pin Groups** window.
2. Select the appropriate pin group on the **Pin Group List** area.
3. Click **Edit Pin Group** to open the **Pin Group Editor** 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. Select unwanted pins and click **Delete Pins** to remove them on the group.
5. Click **OK** to close the window.
6. Click **Close**.

Renaming a Pin Group

1. Open the Components window.
2. Click **Pin Group** > *GroupWithOldName*.
3. From the **Properties** window, click the **Pin Group** value.
4. Enter the new name of the pin group

Deleting a Pin Group

Pin groups can be deleted in two ways, on the **Components** window or the **Create/Manage Pin Groups** window.

From the **Components** window, right-click **Pin Group** > *UnwantedGroup* and click **Delete**. Alternatively:

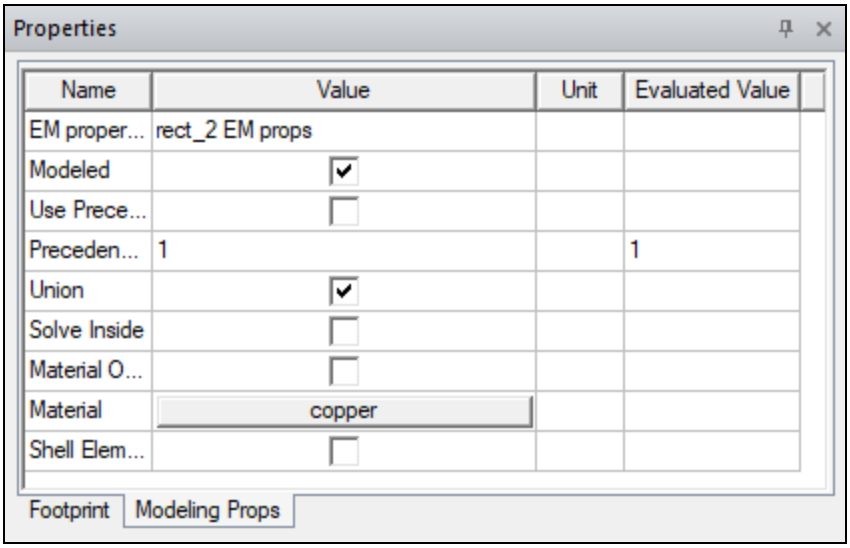
1. Click **Draw** > **Create/Manage Pin Groups** to open the **Create/Manage Pin Groups** window.
2. Select the unwanted group on the **Pin Group List** area.
3. Click **Delete Pin Group(s)**.
4. Click **Close**.

HFSS 3D Layout Properties

Meshing Properties

For most components, the following meshing controls are available under the **Modeling Props** tab of the **Property** window. Select items one by one in the **Properties** window and modify their **Modeling Properties**, as appropriate. To remove the **Modeling Properties** tab on the **Property Window**, click **Draw** > **Remove Modeling Properties**.

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 following:

- **EM properties** field — the editable name of the primitive.
- **Modeled** check box — specifies the object is meshed when the circuit is modeled.
- **Use Precedence** check box — specifies the mesher examines the **Precedence Value** when deciding object precedence.
- **Precedence Value** field — when **Use Precedence** is checked, enter any integer. Where polygons from different layers/materials overlap, the mesher decides precedence on the basis of Precedence Prioritizing.
- **Union** field — merges overlapping polygons into a single object.
- **Refined** check box — causes the object to be refined. When unchecked, the object are not refined.
- **RefineFactor** field — enter a positive integer or variable value that specifies the level of refinement. The larger the value, the greater the refinement. A value of one specifies the same refinement relative to other objects. One-half specifies half as much refinement relative to other objects, and two specifies twice as much refinement. If a zero value is entered, the **Refined** area is automatically unchecked. If a value less than zero is entered, the value is automatically changed 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.

> **Note:**
>
> Data with different meshing characteristics cannot overlap. Overlapping objects must possess the same **RefineFactor** and **NoEdgeMesh** values. If the values differ, the following warning message is displayed: “Data on Layer trace intersects data on the same layer with different properties.”
>
> A warning message is also displayed if self-intersecting components yield unexpected results. Self-intersecting polygons and voids are allowed, but they may not mesh as expected.

- **NoEdgeMesh** — when checked, specifies that no special rectangular-edge mesh is used during modeling.
- **Solve Inside** — when checked, enables the HFSS **“** solve inside**”** option for that particular mesh body. All intersecting mesh bodies should have a common set of modeling properties. If a touching polygon does not have “solve inside” set, the attribute is ignored (and a warning is issued).
- **Material Override** — specifies a material override for the HFSS solver. By default, this material is the same as the object or layer the selected object is drawn on. Click the button to open the materials editor. For a new primitive object, the default material is copper.
- **Material** — Click the button to open the materials editor. For a new primitive object, the default material is copper.
- **Shell Element** — assign a volumetric shell element to the primitive object.

SIwave

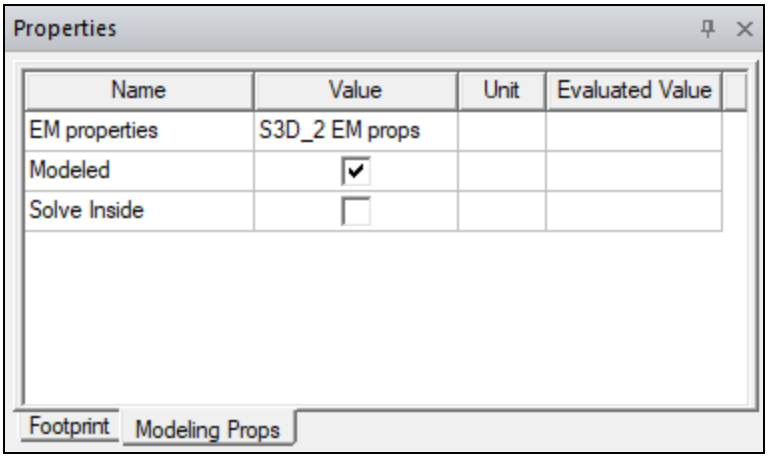
- **DC IR Equipotential Region** — when checked, the DC IR solver uses equipotential regions to force equal voltage across the object. Equipotential regions may be defined as rectangular areas on surface signal layers (i.e., top or bottom).
- **Material** — specifies a material override for the HFSS solver. By default, this material is the same as the object or layer the selected object is drawn on. Click the button to open the materials editor. For a new primitive object, the default material is copper.

> **Note:**
>
> Text should be placed in the symbol layer. Text placed in the signal layer, unless part of the electrical design, must have **Modeling Properties** added with the “Modeled” option unchecked — however, this is not the recommended method and may result in the meshing error “Polygon Meshing Failed”. The preferred method is to locate text in the nearest symbol layer.

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

Modeling Props for 3D Structures

When a 3D structure is selected, only the **Modeled** and **Solver Inside** controls appear under the **Modeling Props** tab of the **Property** window.



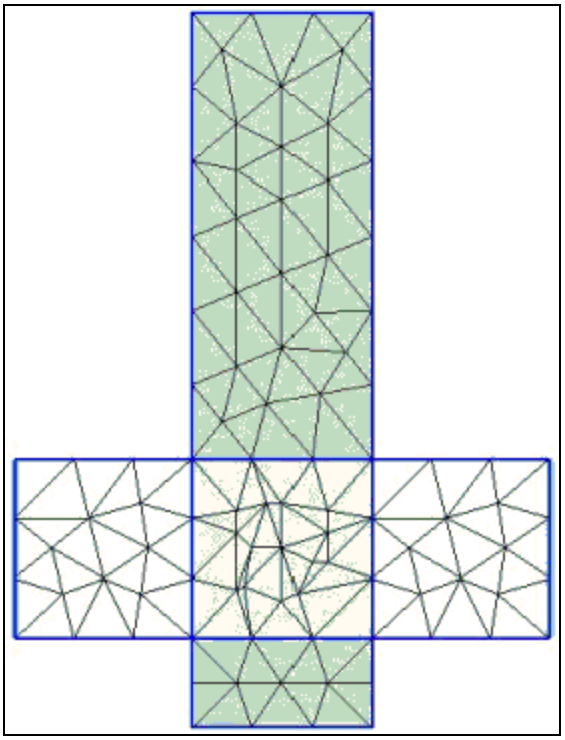
Precedence Prioritizing

Where polygons from different layers/materials overlap, the mesher decides on the material in the area of intersection based on the following rules:

1. For polygons with precedence numbers, the polygon with lower precedence takes priority over those polygons with higher precedence.
2. If the mesher cannot resolve the priority using the precedence attribute, it considers conductivities. The polygon with the higher conductivity takes priority over those polygons with lower conductivity.
3. If conductivities are the same, the stackup index is used. Objects higher in the stackup table (those with a lower stackup index) take precedence over objects lower in the stackup table (those with a higher stackup index).

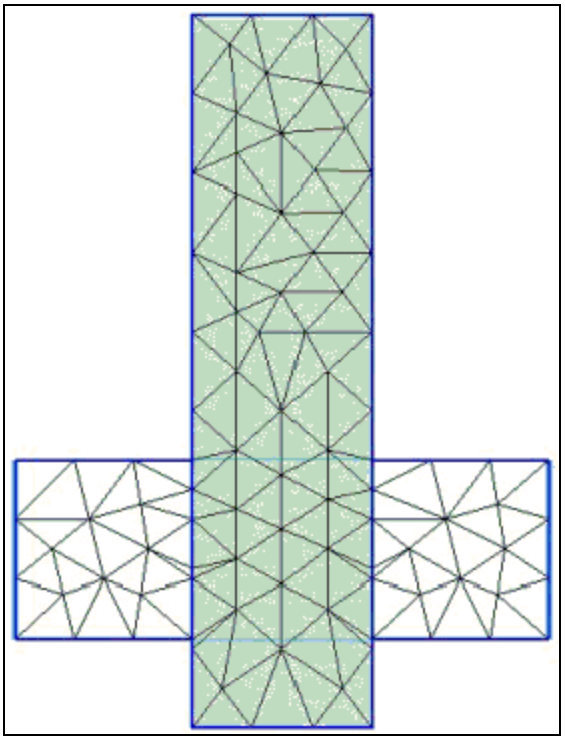
For example, if a blue rectangle (a thin metal) has a lower precedence value than a green rectangle (a thick metal), so its material (blue) takes priority where the two come in contact.

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

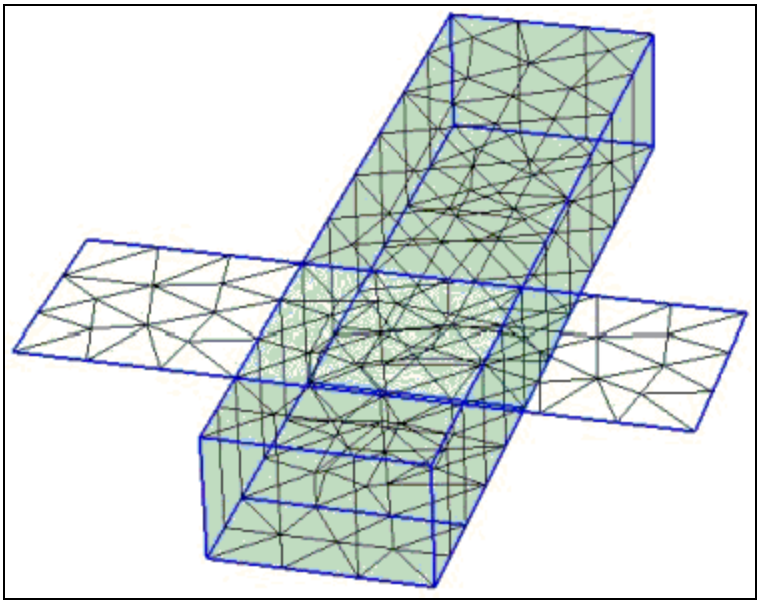


But in this next example, the blue rectangle has a higher precedence value than the green rectangle, so now the green material takes priority.

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



Following is a 3D view of the first preceding example (e.g., where the blue rectangle has the higher precedence value).



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

Thin Film Properties

To define an object as a Thin Film Resistor, select one or more objects in the **Layout Editor** and click **Draw** > **HFSS 3D Layout Properties** > **Convert to Thin Film Resistor**. This defines the object(s) as a resistor and adds the **Modeling Props** tab to the **Property Window**:

The following meshing controls are available under the **Modeling Props** tab of the **Property** window:

- **SheetImpedance** specifies the object impedance value

Select items one by one in the editor to modify their **Modeling Properties** in the **Property Window**. To remove the **Modeling Properties** tab on the **Property Window**, click **Draw** > **HFSS 3D Layout Properties** > **Remove Thin Film Resistor Property**.

Selecting the Sweep Type

1. From the **Project Manager** window, expand the **Project Tree** and select the active design to make the **HFSS 3D Layout** menu appear.
2. From the **HFSS 3D Layout** menu, select **Solution Setup** > **Add Frequency Sweep** to open the **Edit Frequency Sweep** window.
3. Select one of the following sweep types:

Interpolating Sweep Options (Default)	Generates a solution for an entire frequency range. Best for models when the frequency range is wide and the frequency response is smooth, or if the memory requirements of a fast sweep exceed available resources. All discrete basis solutions are solved prior to interpolating sweeps because it is possible that an interpolating sweep can re-use already solved frequencies from a discrete sweep. An interpolating sweep must be used for Time Domain Reflectometry plots (TDR). When you select interpolating sweep, the **Options** tab is activated. This lets you specify a maximum number of solutions, and other interpolation values.
Discrete Sweep Options	Generates field solutions at specific frequency points in a frequency range. Best when only a few frequency points are appropriate to accurately represent the results in a frequency range.
"Options for Broadband Fast Sweeps" on	Generates a unique full-field solution for each division within a frequency range. Best for moderately-sized

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

page 10-455	models when the frequency range is wide and the frequency response is complicated, where interpolating or discrete sweeps would require too many specific frequency points. When you select Broadband Fast Sweep, the **Options** tab is activated. This lets you specify a maximum number of solutions and error tolerance.

Generating the Surface Current

From the sweep window, select **Generate Surface Current** and the Ansys Electronics Desktop (AEDT)™ electronics systems design platform saves the surface current data at every discrete frequency point. This allows you to plot the current and compute near- and far-field quantities at more than one frequency point. Otherwise, the surface current data is saved at the **adaptive frequency** point of the last adaptive pass, if specified.

> **Note:**
>
> If **Generate Surface Current** is not selected and an adaptive solution is not being performed, surface current data is not saved.

Keep the following things in mind when generating the surface current:

- Gain, axial ratio, far fields, and near fields require the surface current. If you select not to save the surface current and do not perform an adaptive solution, commands associated with these values are not available.
- The **Generate Surface Current** option is only available if you select **Discrete** as the type of frequency sweep. For a Fast Interpolating frequency sweep, the surface current data is only saved at the adaptive frequency. Note that if you are simulating over many frequency points, it is recommended that you do not select **Generate Surface Current**, since the resulting surface current file may become very large.
- For problems with plane waves (HFSS 3D Layout only), the **Generate Surface Current** option is always selected.

Setting up a Full-Wave SPICE Analysis

To determine a suitable frequency range for a solution that are used for a Full-Wave SPICE analysis:

1. Click **Setup Full-Wave Spice** in the sweep setup window to open the **Setup for Full-Wave Spice** 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. Type a value in the **Signal Rise Time** field. This value represents the time scale that characterizes the rate of change of the input time signal to be applied in the circuit simulator. By default, this value is set to 1.
3. Type a value in the **Time Steps Per Rise Time** field. By default, this value is set to 5. The time sampling increment for the entire signal is calculated using: $\Delta t = \frac{\tau}{N_\tau}$ where
   - Δt is the time sampling increment.
   - τ is the signal rise time.
   - N_τ is the number of time steps per signal rise time.
4. Enter a value in the **Number of Time Points** field. By default, this value is set to 500. The input time signal duration is determined using: $N' \Delta t$ where *N* is the number of time points.
5. Click **Calculate**. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform determines the **Maximum Frequency** using: $F_{\max} = \frac{0.5}{\Delta t}$ where F_{max} is the maximum frequency. It determines the **Frequency Step Size** using $\frac{F_{\max}}{N}$ and determines the **Number of Frequency Steps** using N-1.
6. Click **Accept** to transfer the new frequency sweep data to the frequency sweep fields in the sweep setup window.
   - Optionally, click **Cancel** to return to the sweep setup window and maintain previously entered frequency sweep settings.

Guidelines for Calculating Frequencies for a Full-Wave SPICE Analysis

When S-parameter dats is exported on the Ansys Electronics Desktop (AEDT)™ electronics systems design platform to PSpice, HSpice, or Maxwell Spice software, a Full-Wave SPICE analysis can be performed. A Full-Wave SPICE analysis includes full-wave effects in a circuit simulation. If you plan to perform a Full-Wave SPICE analysis, use the **Setup for Full-Wave Spice** window to help determine a suitable frequency sweep range for the solution - one that captures the spectral content of a selected impulse response.

> **Note:**
>
> The frequency sweep ranges suggested in the **Setup for Full-Wave Spice** window box are estimates. You may have a pulse with a wider frequency content and AEDT recommended frequency sweep range may miss some of the high frequencies.

The maximum frequency of the frequency sweep range should be at least five times the inverse of the rise and fall times. If the specified frequency band is too wide, a frequency sweep may have convergence problems. If this happens, try to decrease the maximum frequency until the solution converges.

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 recommended, though not required, the minimum frequency of the frequency sweep be less than the maximum frequency divided by the number of frequency steps. It is usually recommended to have at least 500 frequency steps. A higher number slightly improves the Full-Wave Spice solution accuracy, but also increases CPU and memory requirements to solve the problem. For most cases, using 1000 frequency steps provides a good trade-off between the accuracy and computational requirements.

> **Warning:**
>
> Occasionally, AEDT can fail to solve for the minimum frequency during a Discrete frequency sweep due to a failure of the port solver to converge. If this happens, try to increase the minimum frequency until the solution process completes successfully. However, the minimum frequency should be as low as possible because the low-frequency response determines the steady-state time response.

Clip Planes

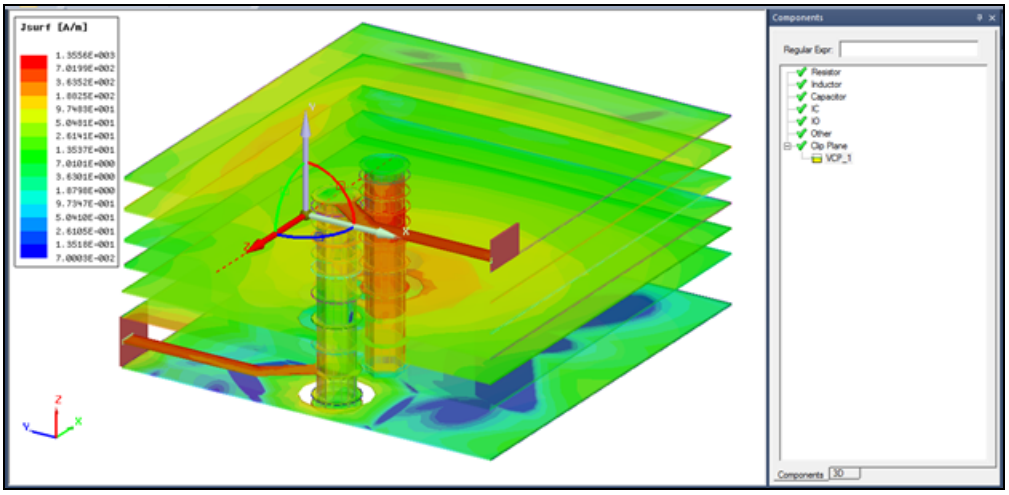
Clip planes can be used to truncate graphics in the Layout view, including mesh and fields plots.

To define a clip plane for the Layout View, click **View** > **Clip Plane**. This places a 3D coordinate-system manipulator into the Layout View. When the clip plane is enabled, graphics are truncated at the XY-plane in the +Z direction.

- To move the clip plane graphically, grab the manipulator's origin and drag/snap to the applicable, position.
- To rotate the clip plane, grab one of the manipulator's origin handles and rotate to the applicable, plane. The incremental angle of rotation can be set in the Toolbars or the shortcut menu's **3D Components** > **Rotation Step** command.
- Position and rotation can also be set explicitly by editing the clip plane's properties directly.

More than one clip plane can be created and, when present, can be applicable, under the **Clip Plane** tree item in the **Components Tab**, are shown in the following diagram.

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



Convert Traces to Planes

Follow these steps to convert traces to planes.

1. Select the trace.
2. Do one of the following:
   - From the **Draw** menu, select **Convert** > **Traces to Planes** or
   - Right-click in the **Layout Editor** and click **Draw** > **Convert** > **Traces to Planes**.

   The trace is converted into a plane. Edit its boundary points.

Convert Planes to Traces

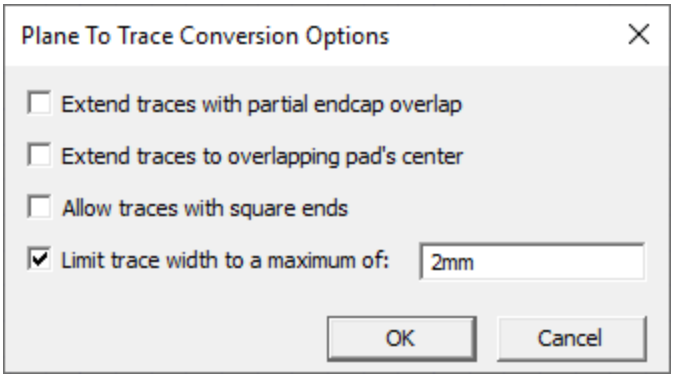
To configure options for converting traces and planes, either:

- From the **Draw** menu, select on the **Convert** drop-down menu or
- Right-click in the **Layout Editor** and select on the **Draw** > **Convert** drop-down menu.

To convert a plane to a trace:

1. Select the plane and navigate to **Draw** > **Convert** > **Planes to Traces Options** to open the **Plane to Trace Conversion 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.



2. Select an appropriate options and click **OK**.

   If the round endcap of a trace overlaps a plane, but the center line of the trace does not intersect the plane, select the **Extend traces with partial endcap overlap** option. This lengthens the trace so the center line touches the bound of the plane and fixes misalignment issues. However if there is other geometry in the small gap that gets filled, it can introduce DC shorts.

3. From the **Draw** menu or the **Layout Editor** right-click **Draw** drop-down menu, select **Convert** > **Planes to Traces**. The selected planes are converted into traces.

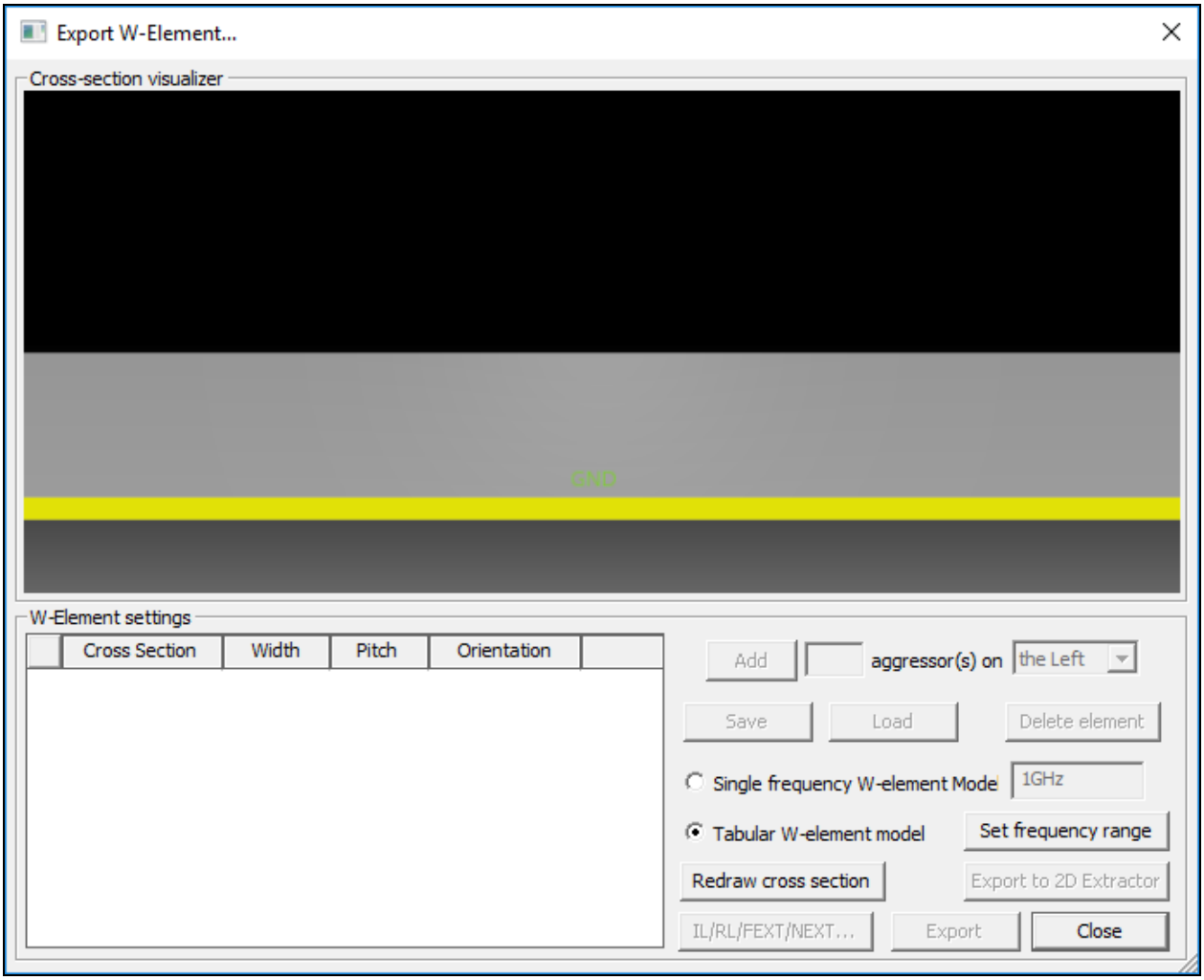
Drawing a Cross-Section for W-element Analysis (HFSS 3D Layout)

The **Export W-Element** window allows you to set up aggressor traces around a victim trace and export the corresponding w-element. Begin by drawing a line in the design area across which the selected cross-section of trace lines are extracted.

To draw the cross-section:

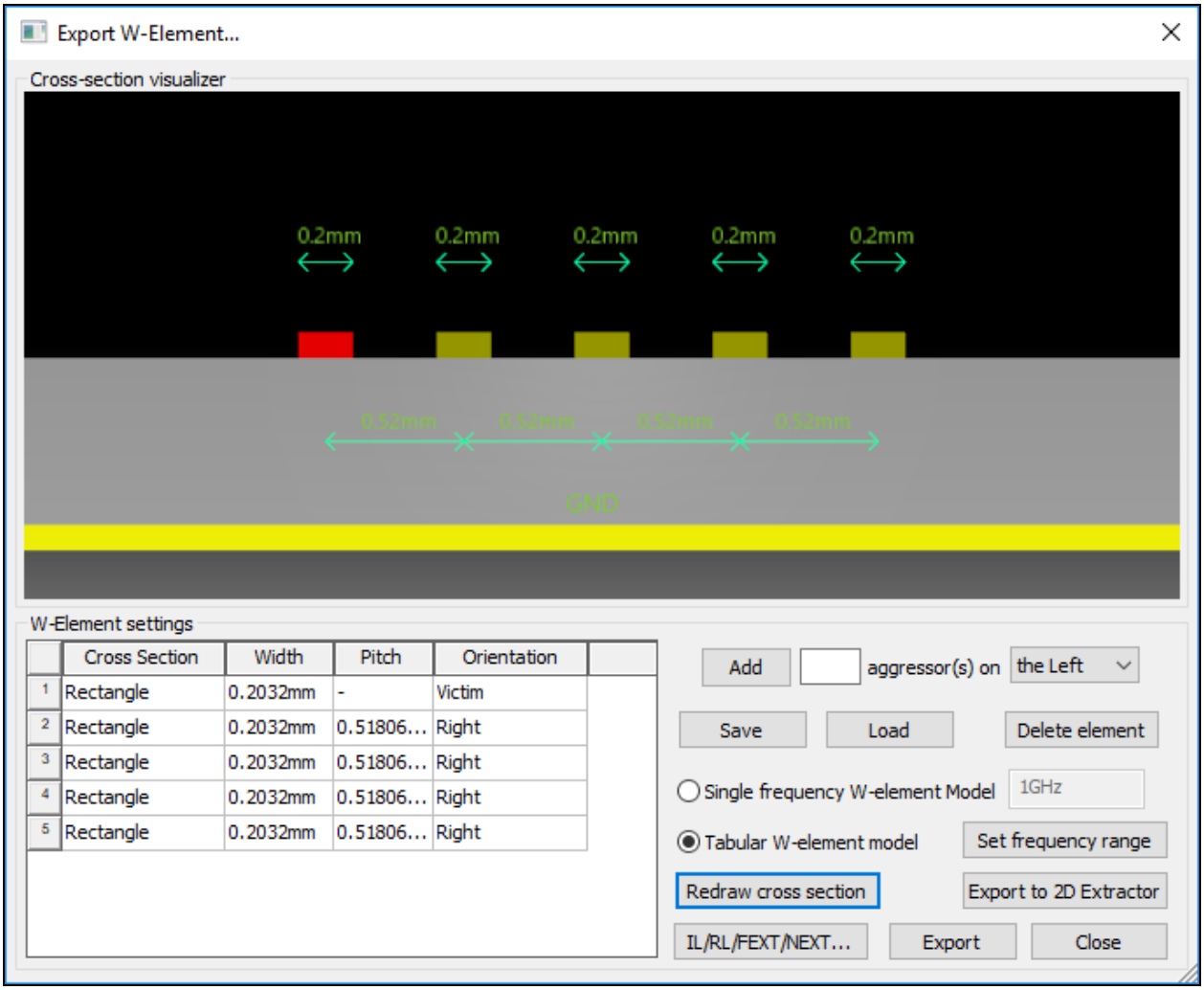
1. Click **Draw** > **Cross Section for W-element Analysis** to open the **Export W-Element** window. The cursor turns into a crosshair.

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



2. Click the location in the design where you want the line to begin. Move the cursor and click again to end the line. The **Export W-element** window updates.

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 an element in the visualizer or select it from the list in the **W-Element Settings** area. The following options are available:
    - **Add** – Enter the number of aggressors to add, and use the drop-down menu to select whether to add them on the **right**, **left**, or on **both sides**. Then click **Add**.

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 previously enabled CPW settings during trace width, impedance, or differential impedance calculation, your options are limited.
>
> - If you selected **Left plane spacing**, aggressors may only be added to the right of a victim trace.
> - If you selected **Right plane spacing**, aggressors may only be added to the left of a victim trace.

- **Delete Element** – Click to remove an element.
- **Single Frequency / Tabular Model** – Select **Single frequency** and enter the frequency in Hz, or select **Tabular** and click **Set frequency range** to set up a range of frequencies.
- **Redraw cross-section** – click this button to clear all information. The cursor transforms into a crosshair again, and you can redraw the line.
- **Export to 2D Extractor** – generates a script to export the selected layer's cross-section to 2D Extractor. Microstrips, stripline and conformal coats are supported. Generating this script launches a prompt asking if you want to run the script. Consult the Q3D Extractor/2D Extractor help for instructions on running an analysis in 2D Extractor.

Also double-click in the **W-Element Settings** area to change width and pitch.

4. Click **IL/RL/FEXT/NEXT** to compute insertion/return loss and time domain crosstalk.
5. Make your selected selections, then click **Export** to open the **Select Spice File to Export** window.

> **Note:**
>
> Selecting **Tabular W-element model** without specifying a frequency range will cause the **Frequency Sweep** window to open. You must specify the frequency range before you can save.

6. Browse to a save location, name your file, and click **Save**. The file is saved in Spice (*.sp) format.

Layout Menu

When a **Layout Editor** design is active, click **Layout** on the menu bar to open the menu for the Layout editor. There are multiple varieties of this menu depending on the type of design being edited (Circuit, HFSS 3D Layout, etc). The following descriptions are a superset of the commands that may appear on the **Layout** menu, though they may appear in different order 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 menu than the order in which they are presented here. For more information, search the Help or use the Help Index to find information on any of the commands that appear on the **Layout** menu.

List — opens the Design List window, which lists the objects in the layout.

Layers — opens the Edit Layers window. Also see Configuring a Stackup.

Nets — opens the **Layout Editor** Nets drop-down menu; the same options are available after right-clicking in the **Layout Editor**.

Push Down — switches the active CS to the next lower level.

Pop Up — switches the active coordinate system (CS) to the next higher level. Relative CSs can be created within other relative CSs to form multiple relative levels.

Place Design — opens the Place Design window.

Connect Pins — connects the selected pins.

Cutout Subdesign — creates a new subdesign from extracted data by cutting out part of a Layout using a selection polygon and/or a net selection. Click subdesign name to rename the new design. The original design is not affected by this operation. See Cutout Subdesign.

Group into Subdesign — creates a subdesign by combining all the selected objects in the current layout. The Project tree shows the subdesign with a default name. Click name to rename the subdesign.

Cross-Probe Schematic — allows you to cross-prob selected schematic and layout elements. When both the **Layout Editor** and the **Schematic Editor** are displaying the same design, select one or more component instances and interface ports in both editors. Select the schematic elements or layout elements to be cross-probed, then select **Cross-Probe Schematic**. For more information, see Cross-Probing Elements and Cross-Probing Selected Elements.

Settings — opens the **Settings** tab of the **Options** window to configure the local option settings in the Layout Editor. See Setting Local Layout Options.

Grid settings — opens the **Display** tab of the **Options** window. The settings on this window control the grid in the drawing area. See Layout Editor Options: Display Panel.

Draw HFSS Air Box — invokes the HFSS Air Box drawing tool.

Draw HFSS Ports — invokes the HFSS Port drawing tool.

Via Styles — opens the **Select Definition** window for **Padstacks**, with the via padstack selected. See Using the Padstack Editor for information on adding or modifying padstack definitions.

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

Pin Styles — opens the **Select Definition** window for **Padstacks**, so that you can add a stackup definition for pins. See Using the Padstack Editor for information on adding or modifying padstack definitions.

Line Styles — opens the Line Styles window that specifies the lines to use on the various layers.

Text Styles — opens the Text Styles window that specifies the fonts to use on the various layers.

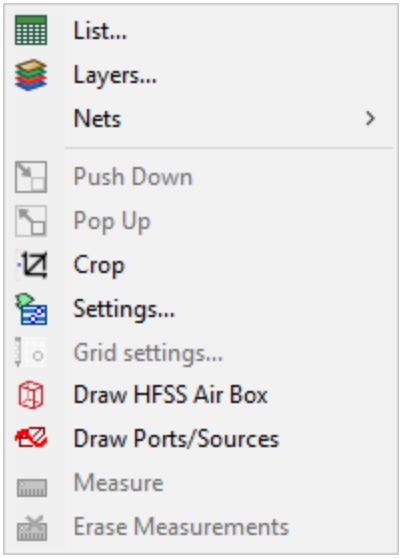
Export to Ansys SIwave™ with ALinks — initiates the export of layout data.

Measure — invokes measurement mode. The cursor changes to a small diamond shape. Click on any two grid-points to display the linear distance between the two points in the current units.

Erase Measurements — clears measurement lines.

IC Mode Commands

The following options appear when the IC mode environment is active. For more information, search the Help or use the Help Index to find information on any of the commands that appear on the **Layout** menu.



List — opens the Design List window, which lists the objects in the layout.

Layers — opens the Edit Layers window. Also see Configuring a Stackup.

Nets — opens the **Layout Editor**Nets drop-down menu; the same options are available after right-clicking in the **Layout Editor**.

Push Down — switches the active CS to the next lower level.

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

Pop Up — switches the active coordinate system (CS) to the next higher level. Relative CSs can be created within other relative CSs to form multiple relative levels.

Crop — enters cropping mode in the **Layout Editor**. See Cropping and Removing Small Fragments in IC Mode.

Settings — opens the **Settings** tab of the **Options** window to configure the local option settings in the Layout Editor. See Setting Local Layout Options.

Grid settings — opens the **Display** tab of the **Options** window. The settings on this window control the grid in the drawing area. See Layout Editor Options: Display Panel.

Draw HFSS Air Box — invokes the HFSS Air Box drawing tool.

Draw HFSS Ports — invokes the HFSS Port drawing tool.

Measure — invokes measurement mode. The cursor changes to a small diamond shape. Click on any two grid-points to display the linear distance between the two points in the current units.

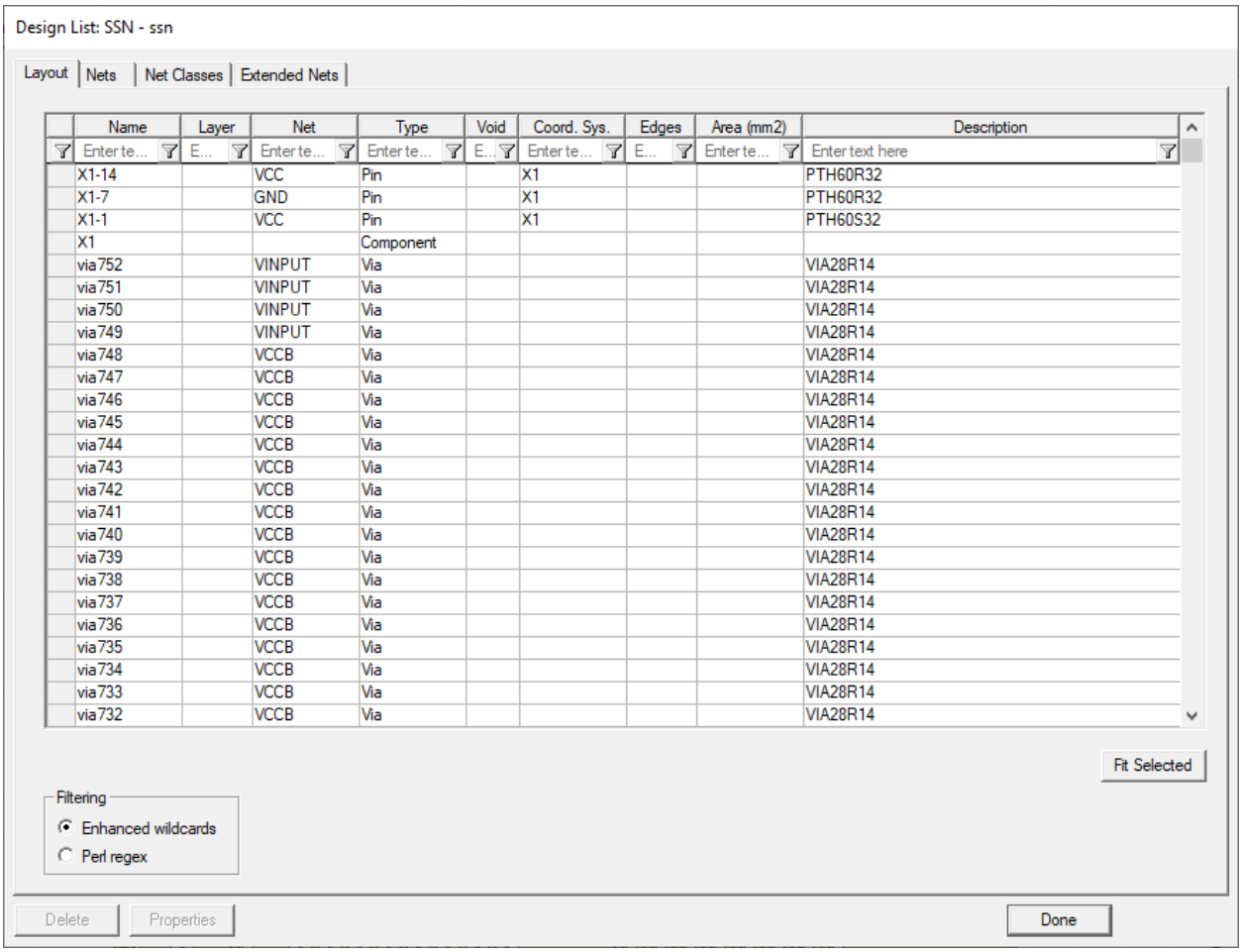
Erase Measurements — clears measurement lines.

The Design List

The **Design List** window lists the objects in the **Layout** window, nets used in that window, net classes, and extended nets. Select **Layout** > **List** on the menu bar 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.

Layout Tab



This tab lists layout objects with descriptive information. On this tab, you can:

- Sort each column by clicking on its title. Click again to reverse the order of the sort.
- Filter a column by selecting the type of filtering you want in the **Filtering** area , then entering a search string in the editable cell underneath the column's title:
  - **Enhanced wildcards** uses the wildcard symbols * and ? to select which objects appear in the list. You can use pattern matching to search a column:
  - * matches any number of characters. For example, **net*** selects objects **net_1a**, **net_1b**, **netLength**.
  - ? matches any one character. For example, **net_?** selects objects **net_1**, **net_2**, **net_a**, **net_b**.
  - **Perl regex** matches a pattern using Perl Regular Expression. You can use pattern matching to search a column:

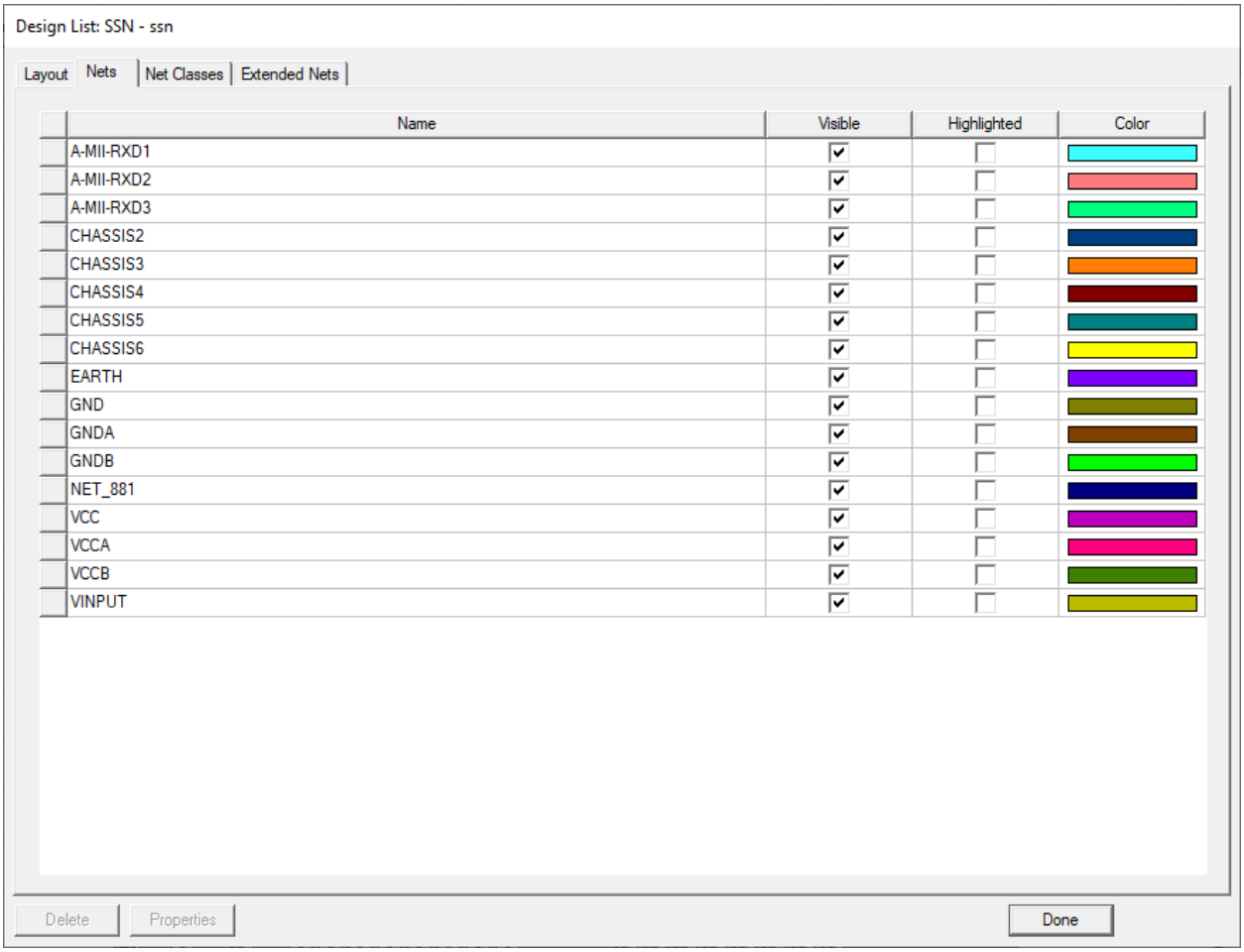
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.* matches any character zero or more times. For example, **net.*** selects objects **net_1a**, **net_1b**, **netLength**.

.? matches any character one or zero times. For example, **net_.?** selects objects **net_1**, **net_2**, **net_a**, **net_b**.

- Click a row to select its corresponding object in the layout. click **Shift** or **Ctrl** to select multiple rows.
- Click **Fit Select** to focus the Layout window on your selected object.
- Click **Properties** to change the properties of selected objects.
- Click **Delete** to remove selected objects on the layout. Warning: there is no confirmation.

Nets Tab



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

This tab lists all the nets in the layout. On this tab, you can:

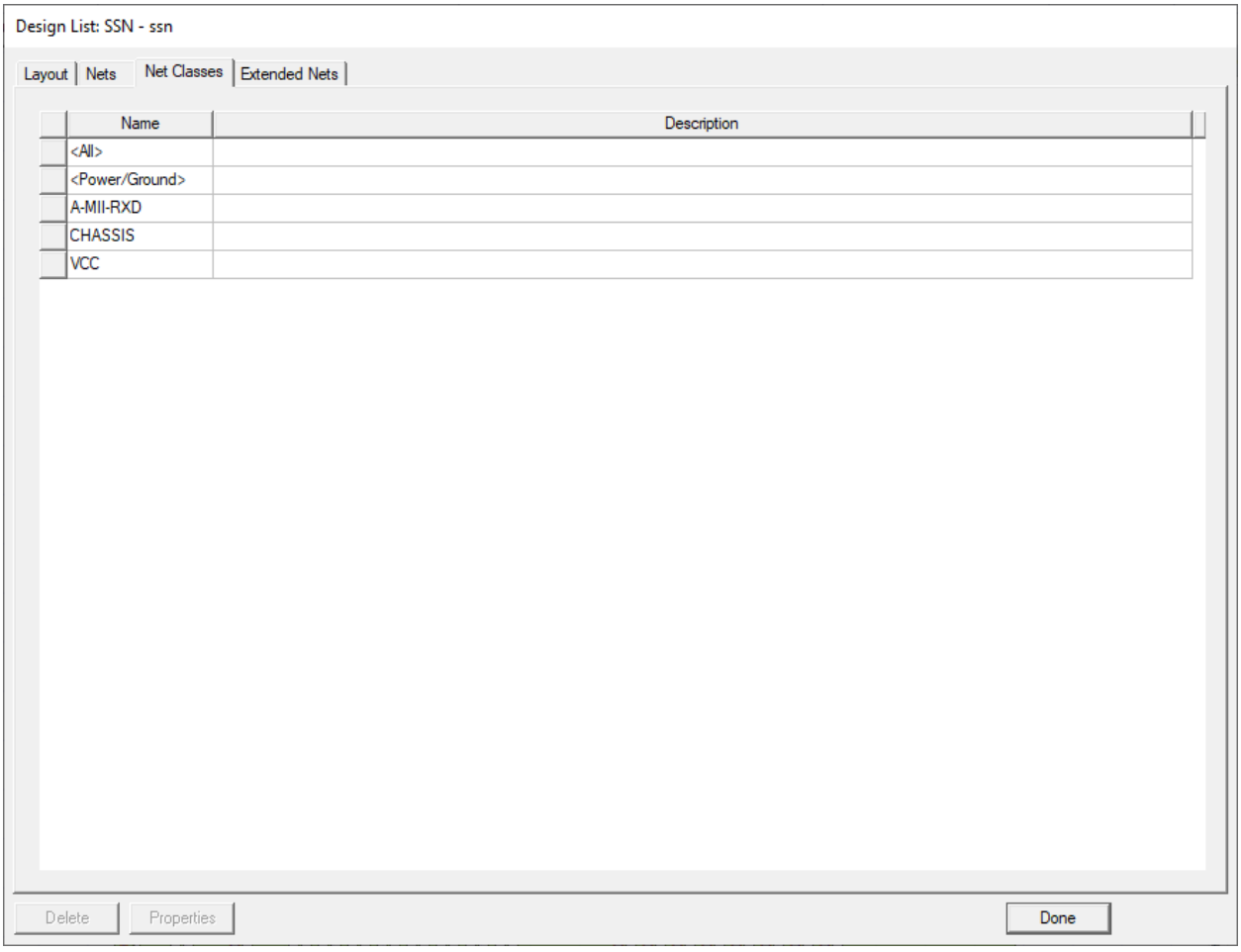
- Show or hide a net by checking or clearing its row's **Visible** check box.
- Highlight the objects in a net by clicking its row's **Highlighted** check box. The objects in that net are drawn in a highlight color. Highlighting is independent of the selection and persists between sessions. Remove highlights from a net's objects on this tab or by selecting a single object in the net and choosing the menu option **Layout** > **Nets** > **Toggle Net Highlight**.
- Identify and change the color of a net. Layouts use color to highlight and distinguish between net, each of which has an assigned color. Double-click a net's **Color** cell to open the **Color** window and select a new color.
- Click a row to select all the objects in that net.
- Remove all items from a net by selecting its row and clicking **Delete**. This does not delete the objects.
- Edit properties common to all elements in a selected net by clicking **Properties** and using the property editor.

> **Note:**
>
> **Shift** +**Ctrl** +selecting an object in the **Layout Editor** selects everything that is in the same net as the object.

Net Classes Tab

The **Net Classes** tab has a list of net classes and their descriptions:

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



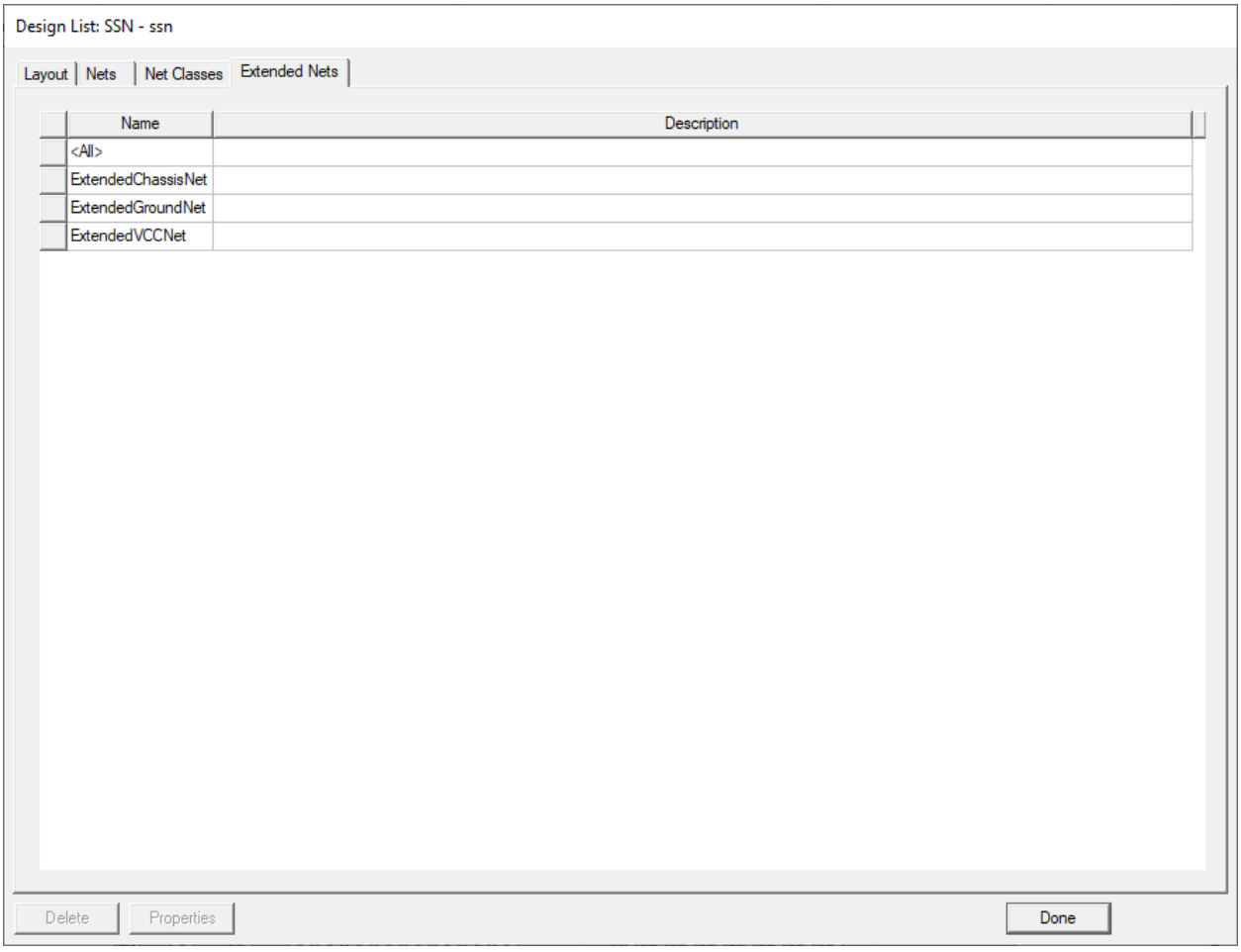
This tab lists all the net classes in the layout. On this tab, you can:

- Click a row to select all the objects in that net.
- Remove all items from a net by selecting it row and clicking **Delete**. This does not delete the objects.
- Edit properties common to all elements in a selected net by clicking **Properties** and using the property editor.

Extended Nets Tab

The **Extended Nets** tab has a list of extended nets and their descriptions:

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



This tab lists all the extended nets in the layout. On this tab, you can:

- Click a row to select all the objects in that extended net.
- Remove all items from an extended net by selecting it row and clicking **Delete**. This does not delete the objects.
- Edit properties common to all elements in a selected extended net by clicking **Properties** and using the property editor.

Nets

Layout > **Nets** opens the **Nets** drop-down menu; the same drop-down menu is also available via the right-click context menu of the **Layout Editor**. Make choices from the following commands to select nets and view net violations.

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 **General Mode**, the **Nets** menu includes: **List**, **Select Physically Connected**, **Select Net Connected**, **Toggle Net Highlight**, **Find Net Violations**, **Show Selected Nets Only**, and **Show All Nets**.
>
> In **IC Mode**, the **Nets** menu includes: **List**, **Select Physically Connected**, and **Find Net Violations**.

List

Opens the **Design List** window. The **Design List** window lists the name of each net, along with **Color** and **Highlighting** controls. Each net has an assigned color. The color is used when the geometry is colored by net or when a net is highlighted. Each net has a check box to activate **Highlighting**. A highlighted net is displayed opaque when transparency is active.

Select Physically Connected

Selects all primitives, vias and pins physically connected to the currently selected items in the currently active design instance (selections in the **Layout Editor** cannot cross multiple levels of hierarchy). Vias and pins with plated holes are followed across layers. Connectivity through primitives in subcircuits and footprints is also followed (unless the subcircuit is a blackbox, in which case it owns its own independent stackup).

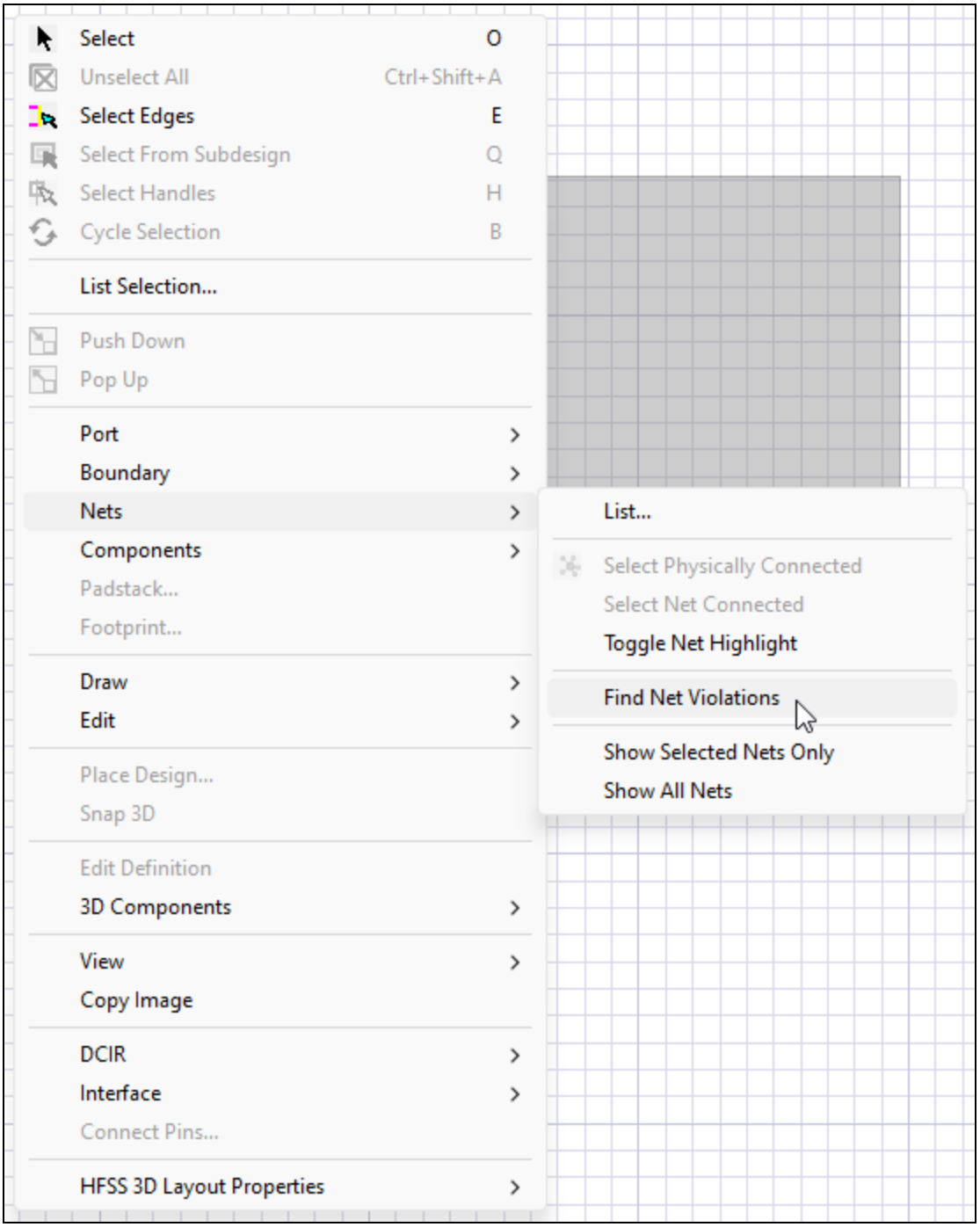
> **Note:**
>
> The starting object cannot be an integrated circuit subdesign.

Find Net Violations

Identifies net violations that occur after a user modifies the physical layout in the **Layout Editor**. The tool updates dynamically and checks whether any object is touching a net it should not.

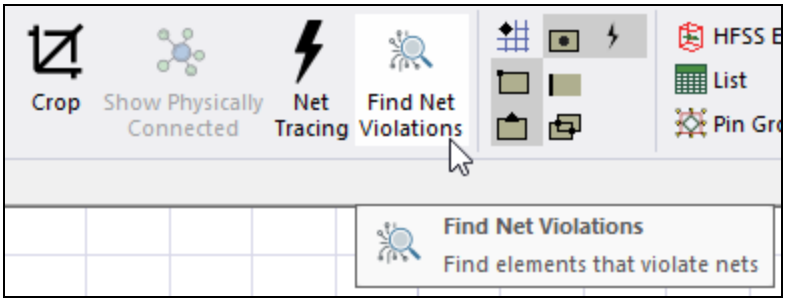
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To use the net violations tool, do either of the following:
   - From the **Layout Editor**, right-click and select **Nets** > **Find Net Violations**.

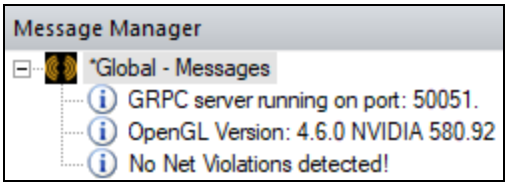


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the **Layout** ribbon, click **Find Net Violations**.

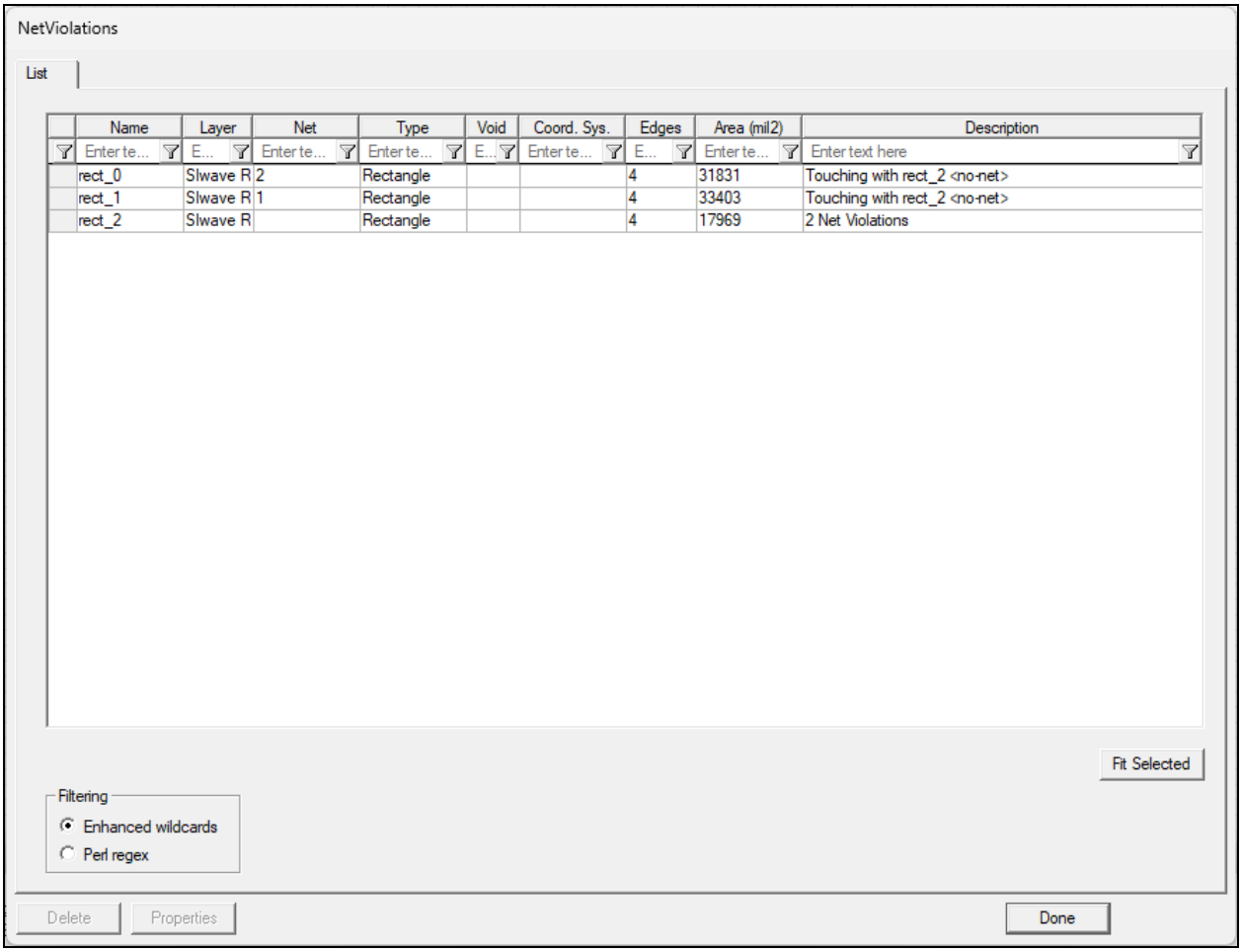


2. If no violations are detected, a message appears in the **Message Manager** window.



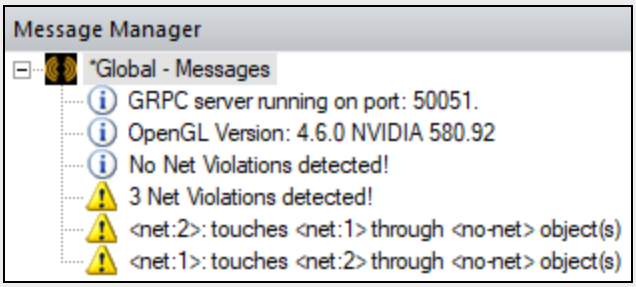
3. If one or more violations are detected, the **Net Violations** window opens. The **List** tab displays components with net placements and violations, shown in the **Description** column.

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



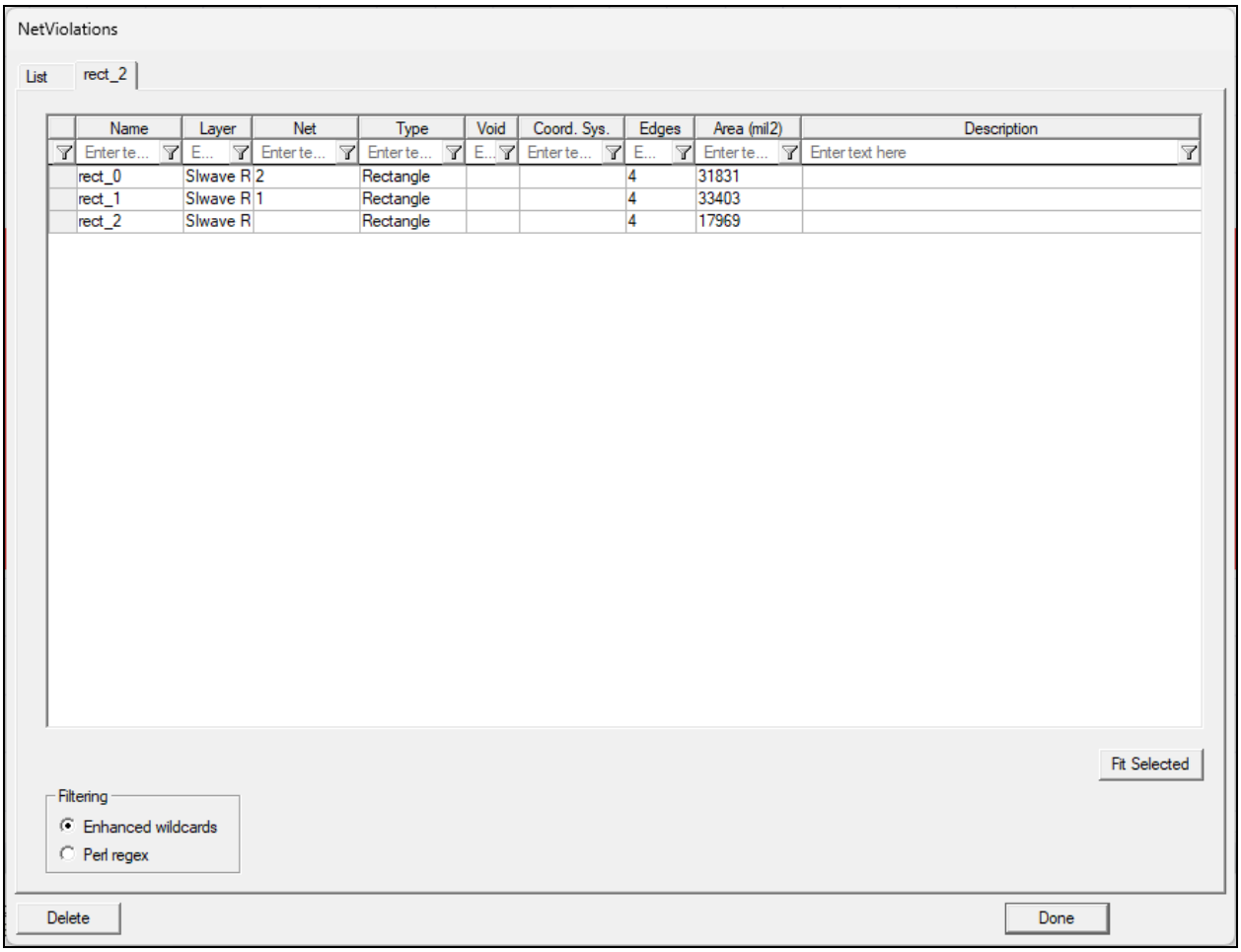
Note:

The **Message Manager** window also displays the number of net violations and a list of the specific violations.



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

4. Selecting a component from the list creates a second tab with the component's data and highlights the component in the **Layout Editor**. Click **Fit Selected** to fit the view over the selected component.
5. Selecting the new tab reveals that component's specific connection data.



6. From either tab, components can be selected and then removed by clicking **Delete**.
7. Click **Done** to close the **Net Violations** window.

General Mode Only

Select Net Connected

Selects all items on the same nets as the currently selected items.

Toggle Net Highlight

Toggles the highlighting for all objects in the same nets as the currently selected items.

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

Show Selected Nets Only

Hides nets that are not selected.

Show All Nets

Makes all nets visible.

Cutout Subdesign

Cutout Subdesign can be used to filter nets and/or clip a design by a polygon. To process the operation in the active design, select **Use Current**. To create a new design, select **Create new** and select whether the result should be created as a sub-design.

- If a polygon is selected before the operation is performed, it becomes the default extent polygon.
- If the selection is made entirely by net, then no geometry should be selected before invoking the operation. If multiple polygons are selected, the union of these polygons forms the extent.
- If no data is selected, a new design is created with a stackup identical to the source design, but no geometry is transferred. Note, however, that **filter geometry by net** must also be selected.
- **Cutout Subdesign** works on hierarchy and footprints, however, everything is **flattened** into the sub-design that is created.

Select **Layout** >**Cutout Subdesign** to open the cut out window:

- **Destination parent design** — select "Use Current" to process the cutout in the active design, or choose an existing top-level design from the drop-down menu, or use "Create new parent Circuit Design" or "Create new parent EM Design". Only top-level designs with a stackup that matches the source design appear in the drop-down menu. If a new design is created, it takes its name on the original source design.
- **Create the cutout as a sub-design in the parent** — when selected, data is copied to a new sub-design within the parent; otherwise, the extracted data is inserted into the top-level design.
- **Filter geometry by net** — when selected, the net data is used to control the data selection and clipping; otherwise, the data in the net table is completely ignored.
- **Nets**:
    - **Net** — the name of the net.
        - **<no net fill>** — refers to polygon data without a net association.
        - **<no net trace>** — refers to trace geometry (wide-line or plugin) without a net association.
    - **Include** — when selected, the net is included in the data transferred.
    - **Clip at extents** — when selected, the net is clipped against the extent polygon.

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

- **Net name filter** — limits the data displayed in the net table. Use the filter to build up a selection of nets. The filter has no affect on the final outcome of the operation (e.g., if "Y" is entered, only those nets beginning with the letter "Y" are displayed; if "C*0" is entered, then all nets matching the wild-carded pattern are displayed). Nets that are not visible retain their "include" and "clip" settings.
- **Extent polygon** — name of the extent polygon.
- **Reset** — resets the extent polygon back to the original polygon; if there was no original polygon, then reset clears the auto-generated polygon.
- **Create a sub-design for the included region** — only applicable if you are creating a sub-design in the top-level design and after an extent polygon has been defined. This option represents the typical use of the Cutout Subdesign tool. The sub-design inserted into the top-level design consists of the data contained in the extent polygon (and filtered by the net selection). All geometry in the nets that is designated as "clipped" are truncated at the boundary of the extent polygon.
- **Create a sub-design for the excluded region** — same as the previous option, however, the sub-design contains the data outside of the extent polygon, filtered by the net selection and clipped as appropriate. It is possible to select both options; in which case two sub-designs are created: one of data in the extent and the other of data outside of the extent.
- **Auto Generate Extent** — displays a second window that permits one to create an extent polygon derived either from the non-clipped nets selected or the current extent polygon. The auto-generated polygon is displayed in the Layout (though it may be difficult to see). It is possible to execute this option as many times as one wishes, adjusting the parameters as needed.
    - **Data source** — the auto-generated extent can use either the current polygon selection or the non-clipped nets as the basis for generating a new extent polygon.
    - **Expansion Type** — the derived region can be defined as a bounding box, a conformal polygon, or a convex hull of the selected data source.
    - **Expansion** — the auto-generated extent polygon is an expansion of the selected data source; the expansion is either a factor (unitless fraction) of the overall source polygon extent or an absolute offset (e.g., 1mm, 20um).
    - **Cornerstyle** — the type of corner to insert in the expanded polygon. Where a "corner" (the simple intersection of the offset edges) can result in rather poor results if there are acute or near-acute angles in the source polygon.
    - **Expansionincrements** — at times, a single, large offset can generate a poorly formed polygon (a preview of the resulting polygon can be observed in the Layout window). The suitability of a polygon is entirely subjective. Improve the general quality of a polygon by breaking up the expansion into a series of steps (increments of 2, 3, 4, etc.). Such increments represent the number of iterations used to reach the full expansion distance. To improve the general quality of a polygon, the number of increments need not be large; you can start with a small number and increase them as appropriate. However, it is important to understand that increasing the number of iterations also increases the time to compute the expansion. So

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

obviously, a very large number of iterations has a significant impact on computation time.

Line Styles

Line Styles opens the **Line Styles** window box. Use the radio buttons to select a line style:

- Click in the **Width** field for a line style to change the width for that style, including the units. Press **Enter** to complete the change.
- Click in the **Bend Type** field to change the bend type (Corner or Round). Press **Enter** to complete the change.
- Click in the **Cap Type** field to change the cap or line-ending type (Flat, Extended, or Round). Press **Enter** to complete the change.

Since a line has nonzero width, its CapType determines how its ends behave:

- **Extended** — extends the line by its own width past the endpoint you click.
- **Flat** — cuts the line perpendicularly at the endpoint you click.
- **Round** — ends the line with a semicircle.

The existing technology files define line/trace styles, including a CapType (typically Flat) for each style. If you select None as the technology type, you'll automatically get a palette of four styles with a variety of widths, BendTypes and CapTypes.

- Click **Add Style** to add a new line style to the table.
- Click **Delete Style** to remove the selected style on the table.
- Click **OK** to save your changes, or **Cancel** to close the window without changing anything.

Legacy Line Style Attributes

The attributes "Corner" and "Round" appear in the BendType menu. The legacy attributes "Mitered (Legacy)" and "Curved (Legacy)" also appears if present in the design.

APDPath

- The "Round" BendType is the same as the bend type used in APDPath.
- The legacy APDPath is replaced by a line with a "Round" CapType and “Round" BendType.
- The APDPaths imported from Cadence APD/Allegro files via **File** > **Import** > **Cadence APD/Allegro/SiP**, and the APDPaths stored in old adsn files, are regarded as lines with "Round" CapType and "Round" BendType.
- To change a path Type to "Cadence APD/Allegro:APDPath", change the path Type to "line". Then set the CapType and BendType to "Round".

Scripting

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

- Any scripting command that converts any kind of path to APDPath, converts the target path to a line with "Round" CapType and "Round" BendType.
- Any scripting command that converts Bend Type of a line to "Mitered" or "Curved", converts the BendType of target line to "Corner".

Specifying Line Ending Types

HFSS offers independent CapType settings for specifying the start and end of lines with new trace rendering.

- There are two separate properties for specifying the ends of a line: "**StartCapType** " and "**EndCapType** ".
- You can set **EndCapType** as "Same as Start", and the **EndCapType** is binded to **StartCapType**.
- If you change **StartCapType** to Extended, the end cap type also changes to Extended.

Text Styles

Text Styles opens the **Text Styles** window. The window specifies the fonts to use on the various layers. The following subsections provide guidelines to use when assigning a font.

System and Plotter Layers

A font must support mirroring, rotation, and scaling to be capably rendered on a photoplotter. There are two categories of layers as they relate to the ability to plot a font.

System Layers

System Layers text does not need to be plottable. The stackup, service, and user layers are system layers.

Plotter Layers

Plotter Layers text must be capable of being plotted. All layers that are not system layers are plotter layers.

Stroke and TrueType Fonts

There are two categories of fonts:

- **Stroke** — (Hershey) fonts can be mirrored and rotated. A stroke font can be scalable or nonscalable. To indicate a Stroke font, select its **Stroke Font** check box in the **Text Styles** window.

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

- **TrueType** — (Win) fonts cannot be mirrored, rotated, or scaled. To indicate a TrueType font, leave its **Stroke Font** check box unchecked in the **Text Styles** window.

Scalable and Nonscalable Fonts

- **TrueType** fonts cannot be scaled (enlarged or shrunk in size). The height of a TrueType font is always specified in points.
- A **Stroke Font** can be scalable or nonscalable. To indicate that a stroke font is scalable, specify its height in the current units (e.g., cm). The height of a nonscalable stroke font is specified in points.

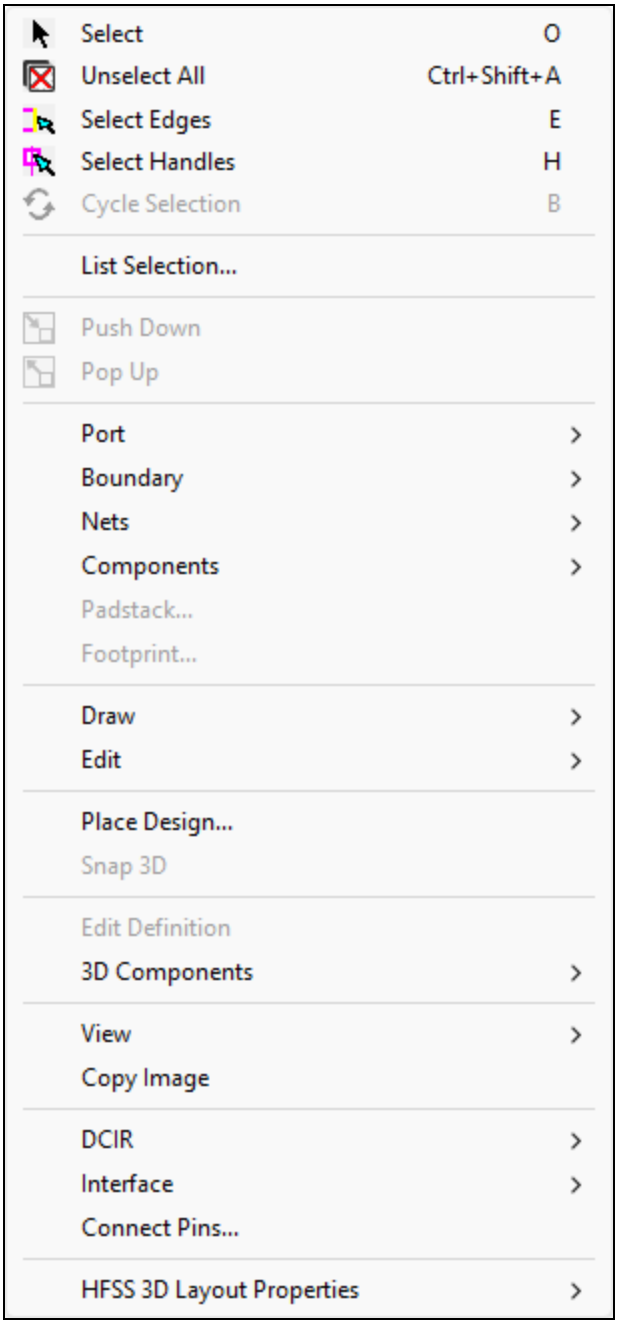
Font Requirements for System and Plotter Layers

- A text style for **System Layers** must be nonscalable (height specified in points). The font can be either a TrueType or a Stroke font.
- A text style for **Plotter Layers** must be scalable (height specified in the current units). Scalability is available only with Stroke fonts. Stroke fonts can also be mirrored and rotated (features which are also required to plot a font.

Layout Editor Context Menu

When a **Layout Editor** design is active, right-click in the design to open the **Context Menu** menu. There are multiple varieties of this menu depending on the type of design being edited (Circuit, HFSS 3D Layout, etc). The following descriptions are a superset of the commands that may appear on the **Context Menu** menu, though they may appear in different order on the menu than the order in which they are presented here. For more information, search the Help or use the Help Index to find information on any of the commands that appear on the **Context Menu** menu for the **Layout Editor**.

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



Select — sets cursor mode to selection of objects.

Unselect All — unselect all objects in the layout.

Select Handles — sets cursor mode to selection of object handles.

Select Edges — sets cursor mode to selection of object edges.

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

Cycle Selection — cycle selection of two or more objects that overlap. Each time you click Cycle Selection, a different one of the overlapping objects is selected.

List Selection — Opens the **Design List** window.

Push Down — switches the active CS to the next lower level.

Pop Up — switches the active coordinate system (CS) to the next higher level. Relative CSs can be created within other relative CSs to form multiple relative levels.

Snap to Nearest Signal Layer — snaps the selected object to the nearest signal layer. Enabled when the active selection is a layout subdesign object.

Port — opens a drop-down menu containing commands for operating on ports.

Boundary — choose to **Create PEC**, **Create RLC**, or **Remove** a boundary.

Nets — opens the **Layout Editor** Nets drop-down menu.

Components — choose to **Create** or **Dissolve** a component.

Padstack — opens the **Edit Padstack Definition** window. Enabled when the active selection includes a padstack.

Footprint — opens the **Footprint Editor**. Enabled when the active selection includes a footprint.

Draw — opens the **Layout Editor** Draw drop-down menu.

Edit — opens the **Layout Editor** Edit drop-down menu.

Place Design — opens the **Place Design** window. Select two components to activate.

Snap 3D — after selecting a 3D component, select to change the component's 3D origin to the electrical connection point (for pins or vias) or the center of the Layout geometry or edge, in the absence of an electrical connection point.

Edit Definition — creates a separate, editable version of the selected component.

3D Components — opens a drop-down menu containing commands for rotating or restricting the orientation of a selected 3D component.

View — opens the **Layout Editor** View drop-down menu.

Copy Image — create a global copy of the selected objects on the clipboard for pasting into a different application.

DCIR — opens a drop-down menu containing DCIR elements to add to a design (i.e., **Voltage Source**, **Current Source**, **Terminal** and **Voltage Probe**). The same elements are available from the **Layout** ribbon.

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

Interface — opens a drop-down menu containing interface ports and ground point to add to a design (i.e., **Create Interface port**, **Create Ground**, **Add Interface Ports at Unconnected Pins** and **Add Grounds at Unconnected Pins**). Interface ports/grounds are also available from the **Layout** ribbon.

Connect Pins — after selecting one component and one subdesign (or a subdesign close to a component with matching pins), opens the Pin Connectivity window.

Activate — causes the activation of one or more selected components.

Deactivate (Open) — temporarily converts the component into an open circuit. Activate restores a deactivated component to the circuit.

Deactivate (Short) — temporarily converts the component into a short circuit. Activate restores a deactivated component to the circuit.

IC Mode Commands

The following options appear when the IC mode environment is active.

List Selection — Opens the **Design List** window.

Port — opens a drop-down menu containing commands for operating on ports.

Boundary — choose to **Create PEC**, **Create RLC**, or **Remove** a boundary.

Nets — opens the **Layout Editor**Nets drop-down menu.

Components — choose to **Create** or **Dissolve** a component.

Padstack — opens the **Edit Padstack Definition** window. Enabled when the active selection includes a padstack.

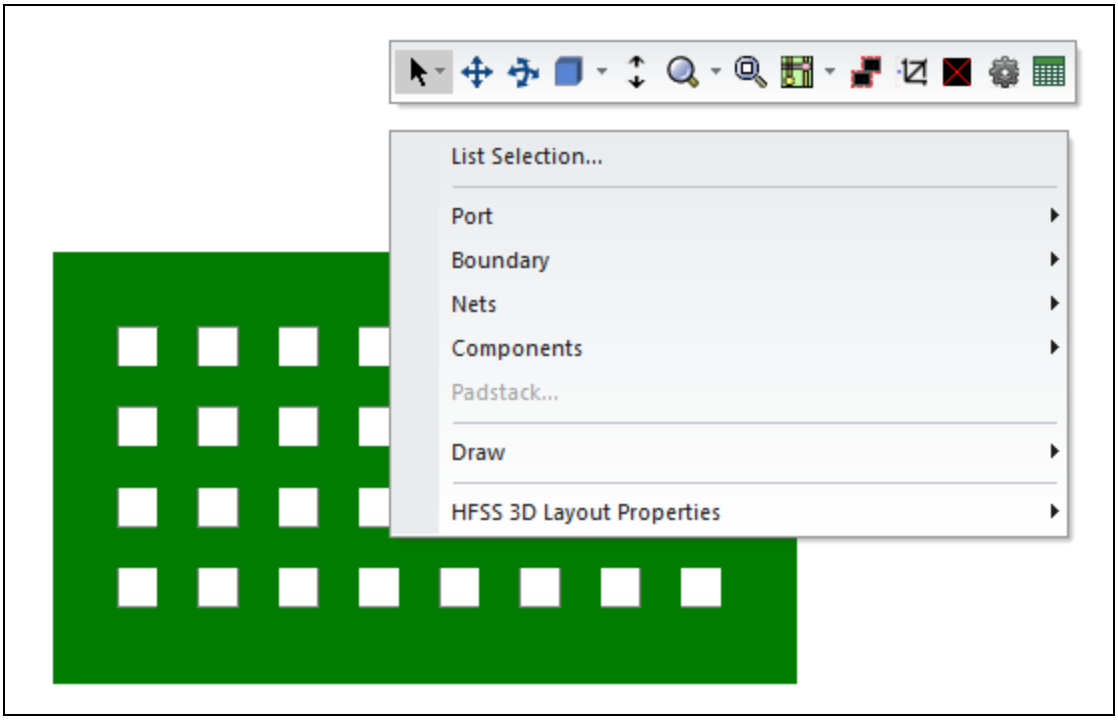
Draw — opens the **Layout Editor** Draw drop-down menu.

HFSS 3D Layout Properties — opens a drop-down menu that allows you to define HFSS 3D Layout Properties.

Shortcut Ribbon (IC Mode)

The shortcut ribbon is unique to IC mode. Right-click anywhere in the Layout Editor to view an additional toolset atop the expected 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 shortcut ribbon provides quick access to a variety of features, which differ depending on whether an element in the **Layout Editor** is currently selected.

Shortcut Ribbon Buttons - No Selection

- **Select** - choose from applicable selection modes (i.e., **Select**, **Select Edge**, **Select All**, **Unselect All**, and **Cycle Selection**). Refer to Selection Modes in the Layout Editor.
- **Pan** - **Click+hold** and drag to move around on the **Layout Editor**. Refer to Panning the View.
- **Rotate** - **Click+hold** and drag to rotate the **Layout Editor** along the X, Y, and Z axes. Refer to Rotating the View.
- **Reset orientation** - reset the current view to several default orientations (i.e., **Top**, **Bottom**, **Left**, **Right**, **Front**, **Back**, **Trimetric**, **Dimetric**, and **Isometric**). Refer to Apply an Orientation to the Current View.
- **Stretch Z** - opens the **Set Layer Thickness Scale Factor** window, which allows users to change layer thickness, or check the **Dynamic** box.

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

- **Dynamic zoom** - choose from several modes (i.e., **Zoom**, **Zoom Area**, **Zoom In**, **Zoom Out**, and **Previous Zoom**), then **click+hold** and drag to scale the **Layout Editor** accordingly. Refer to Zooming In and Out.
- **Fit All** - reset the current view and zoom modes to best view all of the design elements at once. Refer to Fitting All Objects in a View Window.
- **Display Mode** - choose from several display modes (i.e., **Display Default** maintains current transparency settings, **Display Sketch** adjusts to display a wireframe of the design, **Display Solid** maintains the wireframe settings per layer).
- **Wrap Geometry** - all geometry elements are compared to the set **Tolerance**. Every element that meets or falls within the value is wrapped. Refer to Wrap Geometry in IC Mode.
- **Remove small metal islands** - all elements of the design are compared to the set **Tolerance**. Any piece of metal the same size or smaller than the set value is removed from the design. Refer to Removing Small Islands in IC Mode.
- **Settings** - opens the **Geometry Simplification Settings** window, which gives user access to most of IC mode's unique settings. Refer to The Geometry Simplification Settings Window.
- **Layout list** - opens the **Design List** window, displaying all of the designs current primitives (i.e., via the **Layout** tab) and nets (via the **Nets** tab).

Shortcut Ribbon Buttons - Selection

- **Group Vias** - choose from four methods to group the selected vias (i.e., **Persistent**, **Non-Persistent**, **Combine**, and **Dissolve**). Refer to Via Grouping in IC Mode.
- **Snap Vias** - align the selected vias. Refer to Via Snapping in IC Mode.
- **Snap Primitives** - align the selected primitives. Refer to Primitive Snapping in IC Mode.
- **Wrap Geometry** - all geometry elements are compared to the set **Tolerance**. Every element that meets or falls within the value is wrapped. Refer to Wrap Geometry in IC Mode.
- **Remove small metal islands** - all elements of the design are compared to the set **Tolerance**. Any piece of metal the same size or smaller than the set value is removed from the design. Refer to Removing Small Islands in IC Mode.
- **Settings** - opens the **Geometry Simplification Settings** window, which gives user access to most of IC mode's unique settings. Refer to The Geometry Simplification Settings Window.
- **Fit All** - reset the current view and zoom modes to best view all of the design elements at once. Refer to Fitting All 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.

- **Zoom to selection** - reset the current view and zoom modes to best view the selected design element.
- **Show Physically Connected Filtering** - every element physically connected to the selected element is immediately highlighted. Refer to Viewing Physically Connected Elements in IC Mode .
- **Erase** - remove the selected element from the design.
- **Flatten Component** - flatten the heiarchy of the selected components.
- **Union** - combines selected polygons, removing internal boundaries.
- **Subtract** - remove all selected polygons except the first placed, as well as all parts of the original polygon that were intersected.
- **Intersection** - remove all selected polygons except for the intersecting region.
- **Duplicate the selected items along a line** - create a copy of the selected component on either the same layer or across multiple layers (i.e., select either **In Layer** or **Across Layers...**.
- **Reconstruct Arcs** - selected polygons are compared to the set **Tolerance**. Every polygon that meets or falls within the value is formed into a circle. Reconstructing Arcs in IC Mode

Layout Windows

The following topics describe the windows available for viewing and editing configuration settings in the Layout Editor.

Merge Layers Window

When a component or object that has a footprint is placed in the layout or in a schematic, HFSS must map the layers in the component object to the layers that have been defined for the design.

- In a mapping, each layer in the component is either merged with an existing layer in the design or inserted as a new layer in the design.
- The mapping is controlled by the **Merge Layers** window. You can specify (with the “Always Show Layer Merging window” Layout Editor option; see Layout Editor Options) the **Merge Layers** window is to be displayed every time a component is placed (the default), or you can have the window display only when mapping of layers cannot be done automatically.
- Automatic mapping can be performed when the design has no layers defined; in this case, all the layers in the component can be inserted as new layers in the design. Automatic mapping is also possible when there is only one choice available for merging layers (e.g., if you have selected the “Top” signal layers as the active layer in the layout , then place a microstrip component (which has only one signal layer associated with it), HFSS maps the object layer to the active layer. If the “Always Show Layer Merging window” option has been deactivated, the automatic mapping is performed and no window appears.).

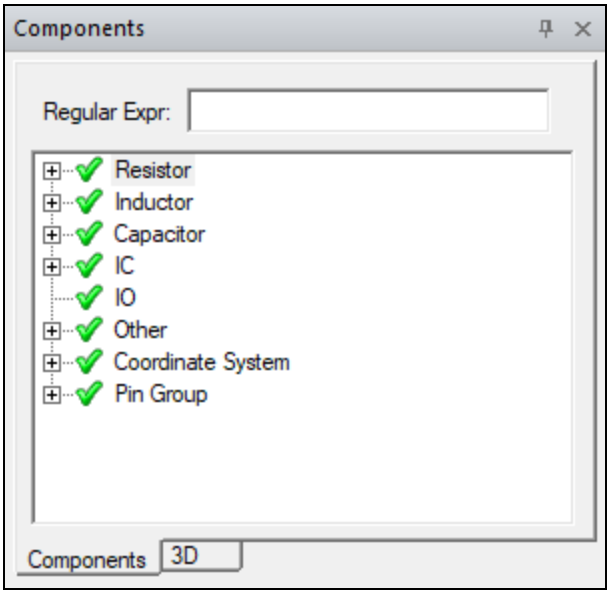
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When automatic mapping is not possible or when automatic mapping has been deactivated by turning on the “Always Show Layer Merging window” option, the **Merge Layers** window appears after a component has been selected but before it can be placed in the layout or schematic.
- The window shows the object layers on the left and the design layout layers on the right. From the preceding example, the object has just one layer (Top) while the layout has three layers (Trace, Dielectric, and Ground).
- When an object layer is adjacent to a layout layer (e.g., the Top and Trace layers), the **M** button is displayed, and the layers are merged.
- When an object layer is adjacent to a blank cell on the layout, the **I** button is displayed, and the object layer are inserted.
- Clicking on an **M** or **I** button toggles the layer between the Merge state and the Insert state. An error occurs if you try to change an Insert layer to a Merge layer when no layers are free for merging.
- **Click+drag** an object layer to a different vertical position. Changing position retains the Merge or Insert state. If an Insert layer is dragged, the layout layers rearrange to leave an empty cell adjacent to the Insert layer. If a Merge layer is dragged, it snaps adjacent to the nearest free layout layer.
- Check the **Flip Source Layers** area to invert the order of the object layers (the order of the layout layers in not changed) (e.g., use this operation can be used to match the layers of a soldered-on object (signal, solder, assembly) on the top to the bottom of a two-sided board).
- When the layer mapping is set up as selected, click **Merge Layers** to execute the mapping. The component is now attached to the cursor for placement in the layout or schematic.
- Clicking **Cancel** cancels the placement of the component.

Working with the Components Window

The **Components** window is a dockable window that is used to view and configure various component settings for Layout components. Use the **Components** window to select a component or a component class, add ports to a component, and configure multiple ports across components. You can also select multiple components in the Layout editor to configure pins and ports for each simultaneously. The window can be resized and relocated.

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



To show or hide the **Components** window on the desktop, click **View** > **Components**. A check box appears next to the command if the **Components** window is visible.

With a component selected, the following configuration controls are available from the right-click menu:

- **Type** – reclassifies the component type to **Resistor**, **Inductor**, **Capacitor**, **IC**, **IO**, or **Other**.
- **Enable**/**Disable** - enables or disables the component.
- **Model** – opens the Set Selected Component Models window, where you can change the component model definition, including die, solder ball, and port properties. Changing the model scope will update every reference designator that belongs to the model definition.
- **Create Ports on Component** – adds ports for the component
- **Remove Ports on Component** – removes ports for the component.
- **Fit Selected** – fits the selected component in the active view window.

> **Note:** When a component classification is selected, any changes are applied to all components within the class.

For more information, see Layout-based Component Encapsulation.

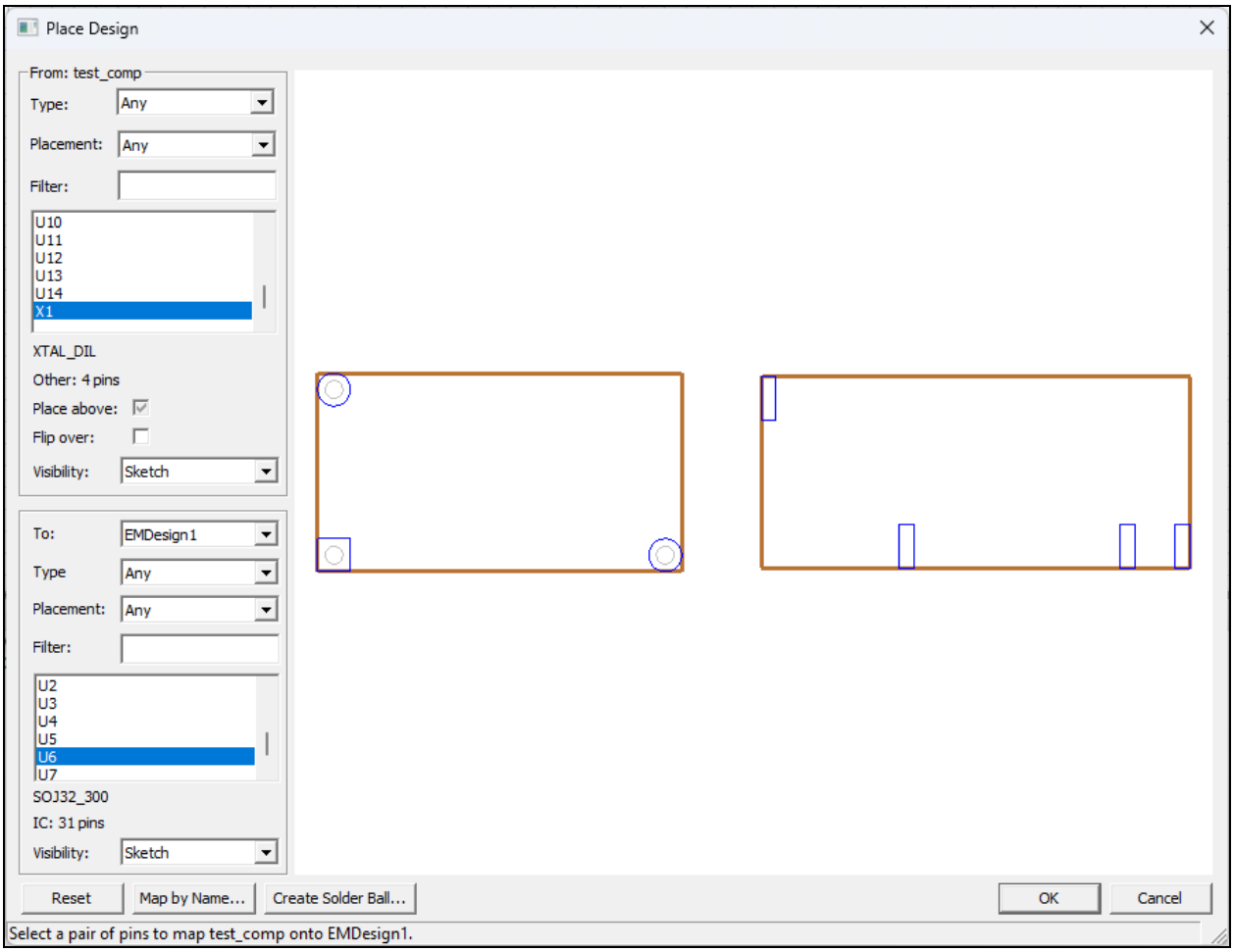
Place Design Window

The **Place Design** window configures how a layout component or a subdesign is positioned within an HFSS 3D Layout design. Use this window to capture or merge the relative positioning

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

of one component or design to another. To open the **Place Design** window, select a component or subdesign along with the component it connects to. Then right-click in the **Layout Editor** and select **Place Design**, or navigate to **Layout** > **Place Design**. The **Place Design** window opens with one or two pairs of pins pre-selected in the preview window adjacent to the parameter drop-down menus.

> **Note:**
>
> If the selected component is fully encrypted and contains exposed components or component groups, the **Place Design** window displays a list of these components/groups. If no component/groups are exposed, the list is empty. Fully encrypted components cannot be edited. Their exposed components are permanent.



Filters and Orientation

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 or subdesign being positioned. The component **To** is what the component/subdesign is being connected to. Both panes contain controls that allow the user to filter and orient components. Make *no* selections to see all components. The following settings are available to configure:

- **Type**: Filter by component type (i.e., **Any**, **IC**, **IO**, **Resistor**, **Capacitor**)
- **Placement**: Filter by placement layer. The list shown is specific to its model. The filter includes **Any** plus all stackup layers in the model (e.g., **Any**, **TOP**, **GND**, et cetera).
- **Filter**: Field where users can filter components by name. Components with the specified **Name**, **Type**, and **Placement** layer are displayed in the list box.
- **Place above**: Controls whether the component/subdesign is placed on and connects to the top layer of the **To** component. The control's default adjusts based on the selected components. This control is not available in the Footprint editor.
- **Flip over**: When checked, this control flips the component/subdesign along the X axis , then rotates it to correctly map the pins of the **To** component. The control's default adjusts based on the selected components. This control is not available in the Footprint editor.
- **Visibility**: Determines how the components are rendered in the window.

> **Note:**
>
> HFSS 3D dynamic link components/subdesigns are not yet linked to components, so a **Port Plane** drop-down menu is shown instead of the component filter controls. A list is displayed showing the port instances in the selected port plane. The **Port Plane** drop-down menu can be used to filter port instances to a specific z plane (or **Any** can be selected to show all port instances). Port instances may be selected in the listbox or graphically selected in the previewer.

Preview Window

A preview of the component/subdesign placement is displayed. As pin pairs are selected, the preview is updated to show the new placement.

Mouse operations are supported similar to the main layout window:

- **Shift**+**click** and drag to move
- **Ctrl**+**Shift**+**click** and drag to zoom in/out
- **Ctrl**+**click** and drag to rotate
- **Ctrl**+**click** to rotate to predefined angles

A shortcut menu on the previewer provides additional 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 **From** component/subdesign can be dragged. When selected in plane, it can also be rotated with the handle that appears. Manual dragging/rotating is only used to assist pin pair selection in the previewer and doesn't actually impact placement once the window is closed.

Reset reverts to the initial preview orientation/placement

Map By Name

Upon opening the **Map by Name** window with the corresponding button, the design is automatically placed by any matching pin pair names (using the full name, net name, and/or suffixes). Users can manually select pin names from the list boxes. The preview in the main window updates as pins are selected in the **Map by Name** window.

Create Solder Ball

Click **Create Solder Ball** to add and configure solder balls to the pins of the **From** component that connect to the **To** components.

Pin Connectivity Window

The Pin Connectivity window is used to connect (or disconnect) components, ECAD and MCAD subdesigns, and symbolic footprints. The Pin Connectivity window is often useful after placing a subdesign with the Place Design window.

Connectables (port instances, pins, and pin groups) are connected between the two selected objects; the first selected object is displayed on the left and the second on the right. When a connection is formed, the left side of the connection's net is updated to the net of the right side of the connection. The net connections are shown as lines drawn on the Rats layer. Connections can be automatically formed by proximity and/or by name. The Place Design window can be used to place a subdesign so that many connections can automatically be formed by proximity.

Basic Usage

1. Open the Pin Connectivity window using one of the following methods:
   - Make two selections (e.g., a subdesign and a component), then select **Connect Pins** on the Layout menu or the shortcut menu
   - Select a subdesign that has already been placed onto a component. The **Connect Pins** window automatically opens if a component is sufficiently close.
2. Manually connect using the right and left arrow buttons:
   - After making a left and right selection, click the right arrow to connect them.
   - Select a grid row and click the left arrow to disconnect.
   - Multiple rows can be selected in the grid to connect/disconnect multiple at once.

Controls

Previewers — Multiple preview panes graphically show the selections being connected:

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 previewer is for the first selection and the right previewer is for the second selection.
- Clicking a port instance, pin, or pin group highlights the corresponding item in the list box or grid (and vice versa).
- Mouse operations are supported in a manner similar to the main layout window:
  - **Shift** +**drag** to move
  - **Ctrl** +**Shift** +**drag** to zoom in/out
  - **Ctrl** +**drag** to rotate dynamically linked HFSS 3D subdesigns
  - **Ctrl** +**click** to rotate to predefined angles for dynamically linked HFSS 3D subdesigns
  - **Ctrl** +**click** to select multiple pins in the component
  - shortcut menu to Zoom In, Zoom Out, or Fit All
- In general, visibility/display settings from layout transfer to the previewer, except:
- HFSS 3D subdesign symbols layer is always displayed.
- 2.5D subdesigns and components are always drawn in solid mode.

List at Left — Shows the left-preview connectables:

- Connectables that are used in one or more connections are displayed in gray.

Connection grid at right — Shows the left-right connections:

- Any connectables in the <Power/Ground> net classification are displayed in red.

Connect by Proximity — automatically connects overlapping connectables:

- Connectables overlapping tolerance depends on the zoom level and the snapping pixel setting. They must also be close in z/layer positions.
- If only a subset of the connections is selected, it may be useful to first click **Connect by Proximity**, then multi-select those rows not selected to be connected and click the left arrow to disconnect them.

Connect by Name — forms connections between left and right connectables that have a matching name or suffix.

Configuration, shortcuts, and additional notes

- Automatic pop-up
- If automatic pop-ups are enabled:
  - When dragging a subdesign onto a component, the Pin Connectivity window automatically appears (if a component is sufficiently close)
  - Place a subdesign in the Place Design window to open the Pin Connectivity window.
  - whether a component is considered "sufficiently close" depends on whether pins/ports overlap.

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, the Pin Connectivity window never appears.
- For new designs, the setting is specified via **Tools > Options > General Options > Layout Editor > Other > Show Pinconnectivity window on drag to matching component**.
- For the active design, the setting is specified in **Layout > Setting > Layout Editor > Other > Show Pin connectivity window on drag to matching component**.
- If the Pin Connectivity window automatically pops up, the "Connect by Proximity" feature is automatically triggered.
- Double-clicking a port instance row connects it to the selected connection grid pin.
- Double-clicking a connection grid row highlights any connected port instance.
- [Net]: the net shows in brackets.
- To disconnect all:
  - From the Pin Connectivity window, select all the rows in the grid (select the first row then hold the shift key while selecting the bottom row). With all grid rows selected, click the left arrow.
  - In layout (outside of the Pin Connectivity window), select a subdesign then click **Draw > Disconnect from Net**.

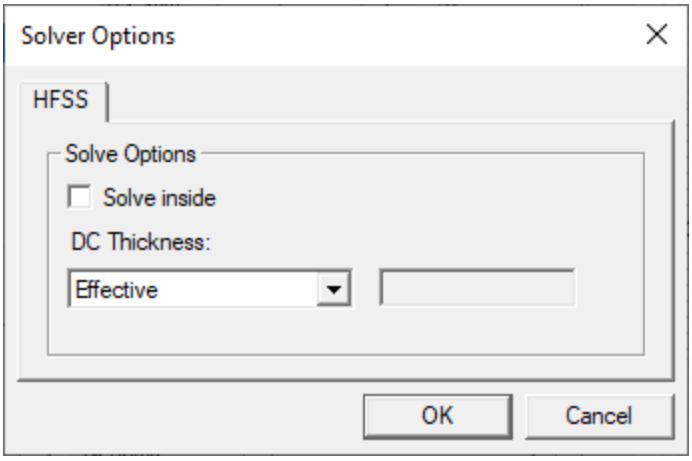
Solver Options Window

Configure solver options on the **Solver Options** window.

Follow these steps to reach the Solver Options window.

1. Navigate to and select **Layout** > **Layers** to open the **Edit Layers** window.
2. Choose **Select signal** from the drop-down menu to enable the options in the **Analysis** area.
3. Check the **Solver** check box to enable **Solver** .
4. Click **Solver** to open the **Solver 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.



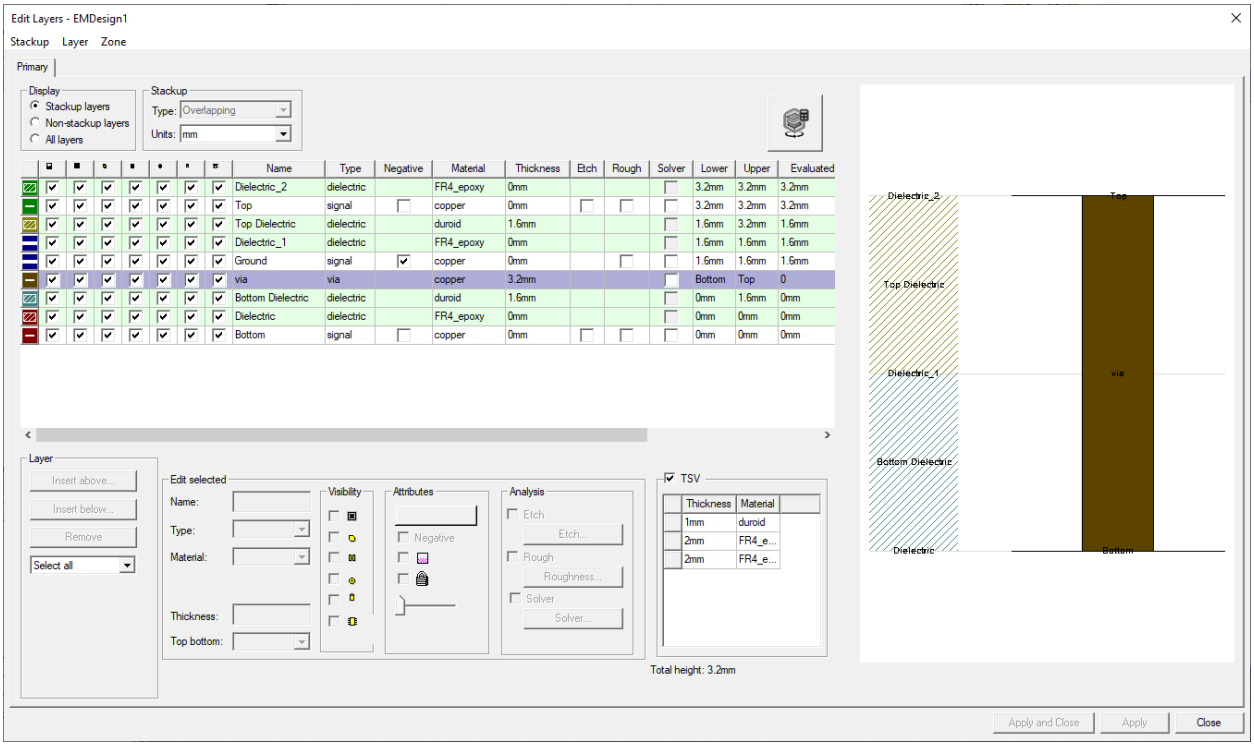
5. Make selections from the following options:

   - **Solve inside** — enables all objects in the layer to have "solver inside" attribute for HFSS solver. It is possible to perform solve inside on an object-by-object basis. See HFSS 3D Layout Properties for more information.
   - **DC Thickness** — Controls how DC thickness is calculated for the conductor lumps on the layer.
     - **Effective**: Calculates as twice the volume/surface area
     - **Layer**: Use layer thickness
     - **Manual**: Use user specified value

The Edit Layers Window

A stackup is a list of physical layers in a circuit board. It can be laminated, overlapping, or multizone. The layers of a stackup are configured in the **Edit Layers** window which contains information about both the stackup layers and the non-stackup (i.e., non-physical) layers. The **Edit Layers** window describes the type, properties, visibility, attributes and analysis of each layer.

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



To open the **Edit Layers** window, do one of the following from the **Layout** tab.

- From the **Layout** ribbon, select **Layers dialog**.
- From the **Layout** or **Footprint** menu, select **Layers** .
- Right-click in the **Layout Editor** and select **Edit Layers**.

The **Edit Layers** window is divided into the following group boxes.

- **Display** – choose which layers appear in the table (e.g., Stackup layers, Non-stackup layers, or All layers).
- **Stackup** – select the Type (e.g., Laminate, Overlapping, Multizone [Beta]) and Units (e.g., centimeters, millimeters, meters, etcera) displayed in the Grid Control Table.
- **Grid Control Table** – view type, properties, visibility, attributes and analysis of each layer
- **Layer** – Insert a new layer above or below, remove a layer, or select either all the Remove, pane adds, removes, and selects types of layers.
- **Edit selected** – After selecting a layer in the Grid Control Table, edit the layer's Name, Type, Material, Thickness, and Top bottom attributes.

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

- **Visibility** – After selecting a layer in the Grid Control Table, edit the layer's Name, Type, Material, Thickness, and Top bottom attributes.
- **Attributes** – After selecting a layer in the Grid Control Table, edit the layer's Name, Type, Material, Thickness, and Top bottom attributes.
- **Analysis** – After selecting a layer in the Grid Control Table, edit the layer's Name, Type, Material, Thickness, and Top bottom attributes.
- **TSV** – Activate and deactivate TSV layers.

> **Note:**
>
> If the **TSV** area does not appear, ensure a via layer is selected or reselected in the Grid Control Table. To create a via layer, refer to **Via Layers**.

- **Stackup Viewer** – Displays a diagram of the layers.

When the **Edit Layers** window opens, it displays the current type of stackup and its most recent display options.

All of the **Edit Layers** window's functionality appears on its **Primary** tab, unless the stackup is multizone. A multizone stackup has a tab for each zone with its controls are spread across the tabs.

From this window, you can import a stackup from an XML file and export an existing stackup to an XML file. Also use the Layer Stackup Wizard.

Stackup Type

AEDT offers three types of stackups in the **Type** drop-down menu:

- Laminate: all layers physically stack one another with no overlap on the Z-axis. This is the default type.
- Multizone: layers physically stack on one another in the Z-axis but each layer can have zones made of different or no material.
- Overlapping: layers can physically intersect one another in the Z-axis.

The **Edit Layers** window offers different controls bases on what the type of stackup. It is generally possible to switch between stackup types. When that cannot happen, the **Type** drop-down menu becomes unavailable.

Display Options and Units

The **Edit Layers** window displays **Stackup layers**, **Non-stackup layers**, or **All layers** in the **Grid Control** pane.

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

Each cell value can be assigned its own unit of measurement. However, selecting a unit of measurement from the **Units** drop-down menu sets the same unit for all cells at once. When the window is first opened, the drop-down menu displays the same units selected in the drop-down menu in the bottom right-hand corner of the **Layout Editor**. If a different unit is selected from the **Units** drop-down menu, the **Edit Layers** window retains the selection the next time the window is opened, rather than returning to the default unit.

> **Note:**
>
> The selected unit affects how layer thicknesses and elevation values are displayed, but does not change the design's underlying geometry.

Grid Control Pane

The **Grid Control** displays the physical and non-physical layers, as determined by the **Display** radio buttons. Layers are color coded:

- Blue: highlighted
- Gray: locked
- Green, light: dielectric layer
- Green, dark: active dielectric layer
- Pink: divider between physical and non-physical layers
- White: signal or non-physical layer
- Yellow: active signal or non-physical layer

A variety of columns allow users to see and edit information about each layer. What can be seen and edited varies by type of layer and stackup. Some columns are hidden by default. Right-click the column labels of the Grid Control pane to see a complete list of columns. If a column does not apply to a layer, it is blank in that layer's row. Thus many columns are empty for non-physical layers.

The Grid Control pane's shortcut menu provides actions for the selected layers. A new layer can be inserted, or the selected layer can be activated. A layer can be locked in place or assigned to a specific zone in a multizone stackup. If multiple dielectric layers are selected, they can be merged.

Grid Control Columns

- **Row Color/Type Icon**: This matches what is shown for the layer in the **Layout Editor**. Clicking this cell to select the row.
- **Wireframe**: Check box to set drawing on the layer as wireframe outlines or solid.
- **Visibility settings**: Controls what is visible on each layer. The visibility settings determine what types of objects are shown on a layer. Similar settings are in the Layers window.

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

    - **Visibility** - Sets visibility for all objects on the layer
    - **Shape visibility** - Sets visibility for shapes on the layer
    - **Line visibility** - Sets visibility for lines on the layer
    - **Pad visibility** - Sets visibility for padstack pads on the layer
    - **Hole visibility** - Sets visibility for padstack holes on the layer
    - **Component visibility** - Sets visibility for components and black boxes placed on the layer
- **Name**: Editable name of the layer
- **Type**: Editable type of the layer. Choices are given in a drop-down menu that varies for stackup and non stackup layers. Ground is no longer a layer type choice. To create what was previously described as a ground or metalized signal layer, set the type to signal and the **Negative** state on.
- **Negative**: Type of geometry to draw on the layer. Geometry on a negative layer cuts away on the layer. Geometry on a positive layer adds to it. This setting is used in conjunction with the signal type to create a layer used as a ground layer. This column is visible if the stackup contains a layer set to negative. Hidden by default.
- **Material**: Editable choice of material for the layer. The drop-down menu contains names of materials already in the project that are allowed for the given type of layer. The first choice in the menu is **Edit**, which allows users to open the **Materials** tab of the **Select Definition** window. Users can type in a project variable that is a text array reference. This allows parameterization of the material (e.g., there is a project array variable $mat that is defined to have the strings "copper" and "Cu_Pkg" and there is a project array $i defined as 1. The user may set the material for a layer to be $mat[$i]. The material used are copper.).
    - **Dielectric Fill** - specifies the material to use to fill in around geometry on the layer. The dielectric material can be selected in the same way that layer material is selected. Hidden by default and not available for overlapping stackups.
    - **Dielectric Constant** - Hidden by default.
    - **Loss Tangent** - Hidden by default.
- **Thickness**: Editable thickness of the layer. It can contain variables or expressions.
- **Evaluated Thickness**: the calculated value of the thickness. Hidden by default.
- **Top Bottom Association**: Allows users to set top bottom association for the layer. Hidden by default.
- **Etch**: Allows etching on signal layers.
- **Rough**: Allows roughness on signal layers.
- **Solver**: Enables the use of specified solver options. Solver options are available for signal and dielectric layers.
- **Lower**: In an overlapping stackup, the lower elevation of signal layers can be user-defined with a number, variable, or expression. The lower elevation of dielectrics is displayed, but not editable.

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

- **Upper**: Contains of the upper elevation of the layer based on the lower elevation and the thickness of the layer. It is not editable.
- **Lower elevation**: The calculated value of the lower elevation. This is useful if the **Lower** value contains a variable or expression.
- **Upper elevation**: The calculated value of the upper elevation. This is useful if the **Lower** value contains a variable or expression.
- **Pattern**: Fill pattern used on the layer. Hidden by default.
- **Transparency**: Transparency values lie between 0 (completely opaque) and 100 (completely transparent). Values may be entered directly into the column or using the **Attributes** slider bar. The slider bar may be used to adjust the transparency for multiple layers. Select multiple rows either directly or through the selection pull-down.

Layer Pane

From the **Layer** pane, you can insert a layer, remove layers, and select layers based on their type. These operations can affect multiple layers simultaneously.

Edit Selected Pane

The **Edit Selected** pane offers controls that can edit multiple layers simultaneously. Select the layers in the Grid Control pane or **Layers** pane first. Controls are only available for the layers to which they apply. A control duplicated by a Grid Control column is always visible, even if its column is hidden. The controls are:

Control	Also available in	Icon or Control
Name	Grid Control column	
Type	Grid Control column	
Material	Grid Control column	
Thickness	Grid Control column	
Top bottom	Grid Control column	
Visibility controls	Grid Control column and Layers window	
Layer color		(color may vary)
Negative	Grid Control column	
Wireframe	Grid Control column	
Transparency	Grid Control column	

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

Control	Also available in	Icon or Control
Locked	Grid Control shortcut menu	
Etch	Grid Control column	
Rough	Grid Control column	
Solver	Grid Control column	

HFSS geometry allows for the presence of an etch factor in the design and modeling of different substrate layers. Check the **Etch** check box to use an etch factor when the mesh is created and check the **Etch** check box to set the etch factor.

By default, layers do not have roughness. To add surface roughness a layer, check its **Rough** column check box. The roughness is added to the layer when its mesh is created. For more information, see Setting the Layer Surface Roughness.

Signal layers may have solver options including Solve Inside Solver. Click the **Solver** check box in the **Analysis** pane to activate the solver options and the Solver button to open the Solver Options window.

Stackup Viewer

The Stackup Viewer is a 2D diagram that displays stackup layers. A column of dielectric layers is on the left with columns of signal and via layers on the right. Via layers are thinner than signal layers. The Stackup Viewer can be customized for useability, to show elevation lines separating signal and dielectric layers, to display only visible layers, and show or hide tool tips and layer names.

Each stratum has a name and color that corresponds to a layer in the Grid Control. Click on a stratum to select its layer in the grid. To show or hide layer names, right-click Stackup Viewer and click **Layer Names**. If layers in the stackup are hidden, they can be shown or hidden in the Stackup Viewer. Right-click the Stackup Viewer and click **Show Only Visible Layers**.

Strata are divided by elevation lines. The elevation lines for signal layers are dotted and for dielectric layers are solid. These lines may overlap depending on the type of stackup displayed. To show or hide the elevation lines, right-click Stackup Viewer, click **Elevation Lines**, then select the type of line.

The Stackup Viewer can be normalized for usability. Normalizing spaces dielectric layers evenly and normalizes the spacing of signal layers relative to the dielectric layers they overlap with. The normalized view emphasizes how the layers stack on one another, instead of the layers' relative size. Remove the normalization to view the layers in their proportional sizes. To use or deactivate normalization, right-click Stackup Viewer and click **Normalize**. You can zoom in on the diagram by using your mouse. To reset the zoom, right-click Stackup Viewer and click **Reset Zoom**.

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

When tool tips are activated, you can hover over a stratum to see its name, relative permittivity, relative permeability, and/or conductivity. To deactivate or activate tool tips, right-click Stackup Viewer and click **Tooltips**. The tool tips only appear if this option is active.

To save an image of the Stackup Viewer, right-click Stackup Viewer and click **Copy Image**. The image is copied to your computer's clipboard.

Color, Visibility, and Patterns

The following topics describe the tools and operations available for configuring color, visibility, and patterns in the **Edit Layers** window.

Setting a Layer Color

When you specify the color of a layer in the stackup, the color is assigned to every object drawn on the layer.

To set the color of objects on a layer:

1. From the Edit Layers window, click the layer.
2. Use the color button in the **Attributes** pane to assign a color to the selected layer.
3. Click **OK**.The color you selected is assigned to every object drawn on the layer.

Setting a Layer Pattern

When you specify the pattern of a layer in the stackup, the pattern is assigned to every object drawn on the layer.

To set the pattern of objects on a layer:

1. From the Edit Layers window, the **Pattern** column is not visible by default; use the shortcut menu in the grid control to make the column visible.
2. From the **Pattern** column, double-click the pattern of the layer you want to change.
3. Select a pattern on the **Select Pattern** window.
4. Click **OK**.The pattern you selected is assigned to every object drawn on the layer.

Setting Layer Visibility

A layer must be visible to be drawn upon. But in each layer, set visibility by individual object types. Perform one of these steps to control layer visibility.

- From the Edit Layers window, use the icons available to select the layers you want to be visible.

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 shortcut menu of the **Layout Editor**:
  - **Set Active**: Makes the current layer the active layer
  - **Show This Layer Only**: Makes the current layer the only visible layer
  - **Show All Dielectrics**: Makes all Dielectric layers visible
  - **Hide All Dielectrics**: Makes all Dielectric layers invisible
  - **Show All Signals**: Makes all Signal layers visible
  - **Hide All Signals**: Makes all Signal layers invisible
- click the **Layout Editor** to alter the following controls for setting visibility by layout-object type.

The following controls are available:

Controls the visibility of shapes

Controls the visibility of lines (paths)

Controls the visibility of pads

Controls the visibility of holes

Controls the visibility of components

Layout Technology

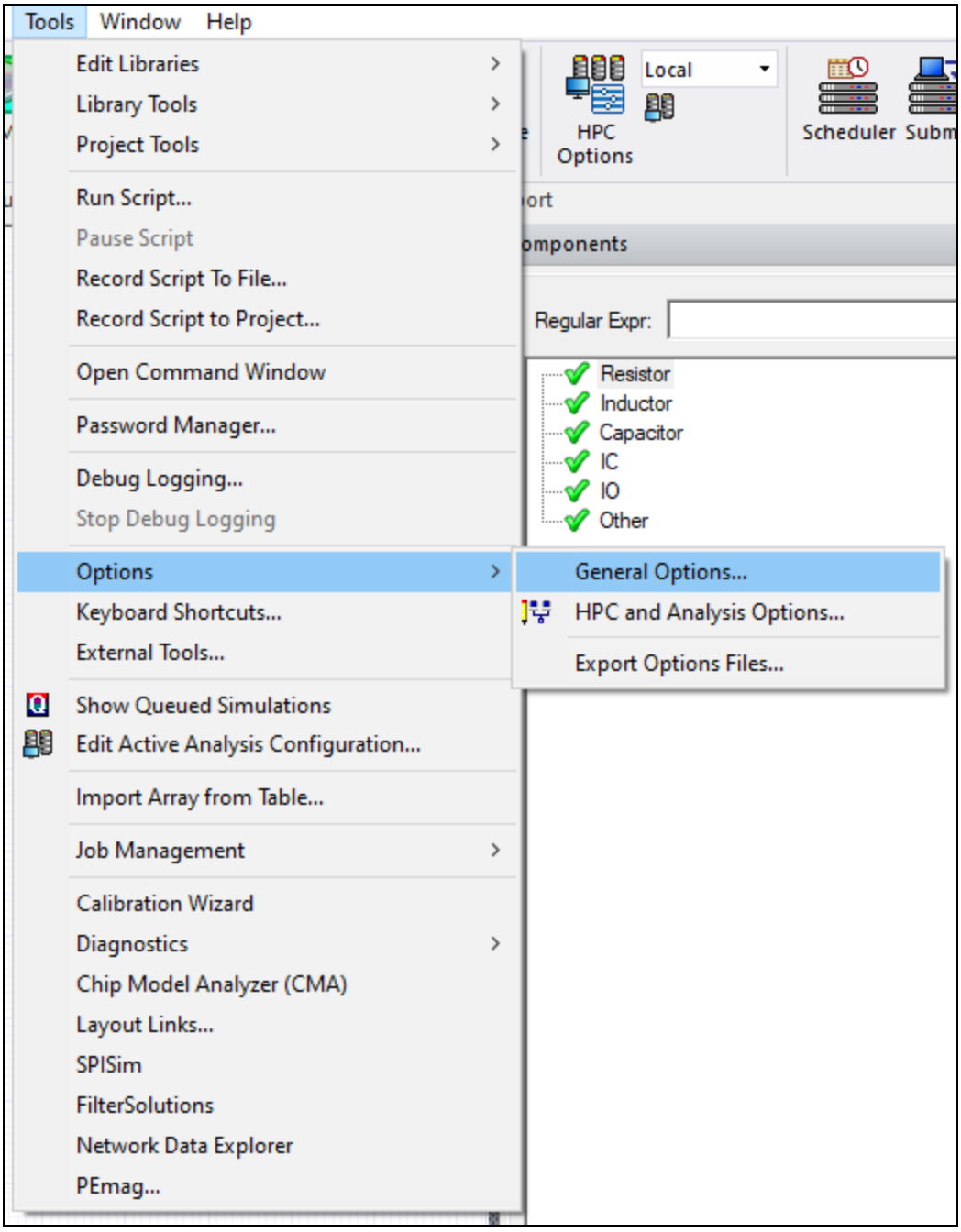
The following topics describe the tools and operations available for configuring layout technology in the **Edit Layers** window.

Choosing a Layout Technology

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform can associate a predefined stackup, (also called the substrate or technology), with each design. A technology includes stackup layers' material properties, units, thickness, elevation, and roughness data, and associated substrate definitions for Nexxim Circuit simulation. The current technology or substrate is applied to components defined for a design or copied in from other designs. Follow these steps to enable the **Choose Technology** window and then select a technology.

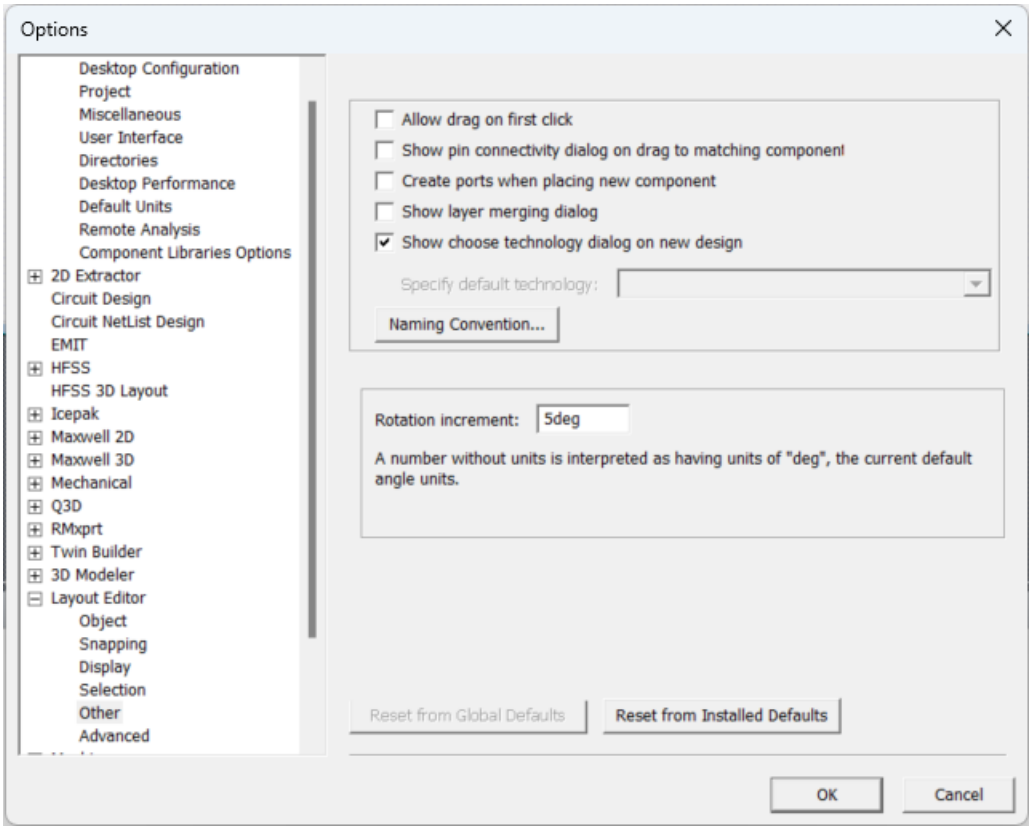
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window.

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



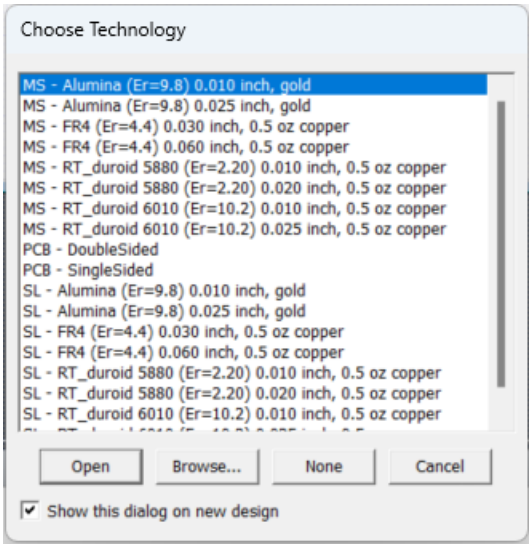
2. Expand **Layout Editor** and select **Other**. Then check the **Show choose technology dialog on new design** box.

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



3. Click **OK** to close the **Options** window.
4. Insert a new design **HFSS 3D Layout** design in a project to open the **Choose Technology** 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. Select the stackup technology on which the design are based. Select a technology, then proceed to design editing without selecting a technology, or abort adding a design to the project.
6. From the **Choose Technology** window, do one of the following:

- To use one of Desktop's predefined technologies, select its entry in the list, then click **Open**.
- To use an existing technology not shown in the list, click **Browse**. Through the explorer window, browse to and select the Ansys Technology *.asty* file, then click **Open**.
- To add a design without choosing a technology, click **None**.
- To select not to add a new design to the project, click **Cancel**.

7. When you specify a technology or click **None**, a blank design of the type you selected opens for editing

   All components that can use a substrate technology automatically assigns the technology you select for the design. This includes components added on the **Components** window and components pasted in from other designs. If no technology has been selected, no substrate is assigned to components.

   As a result, when a schematic from a project that has a substrate definition is copied into another project, the pasted schematic uses the substrate that is defined for the second project, if a substrate has been defined. If no substrate has been defined for the second project, the schematic elements are copied in with no substrate ("SUB" property is empty).

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 import a technology from a technology file, refer to Adding a Technology File to a Design (IC Mode only).

Creating a Layout Technology

You can create your own layout technology and save it for future use. After you have configured a technology in the Edit Layers window, on the **Project Manager** window, expand the **Project Tree**. Then select **[*Active Design Folder*]** and navigate to **File** menu > **Save as Technology File** to open an explorer window that allows you to save the technology in one of the Ansys Electronics Desktop (AEDT)™ electronics systems design platform library directories.

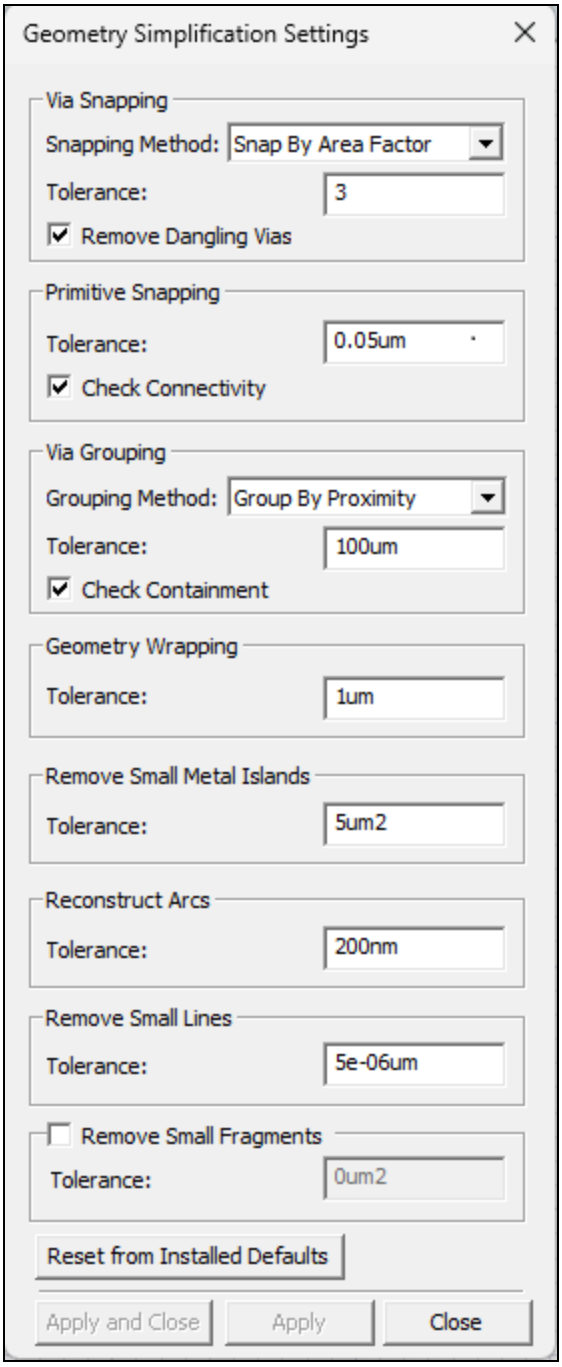
The technology file (.asty suffix) contains the definitions of substrates, layers, materials, padstacks, footprints, symbols, and scripts. The technology file also contains the settings of the Layout options (units, angle units, major and minor grid spacings, snap sources and targets, rotation increment, and hole size) that are in effect for the selected design.

The layout technology you defined is now available for future use and are listed in the **Choose a Layout Technology** window.

The Geometry Simplification Settings Window (IC Mode Only)

IC mode allows users many unique ways to simplify geometry simulations. Refer to the following instructions to open the **Geometry Simplification Settings** window and configure geometry parameters.

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



Opening the Geometry Simplification Settings Window

To open the **Geometry Simplification Settings** window, navigate to the **Layout** tab and select **Settings**.

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 **Geometry Simplification Settings** window, users can adjust the following settings.

Via Snapping

1. Select a **Snapping Method** from the drop-down menu (**Snap By Area Factor** selected by default).
    - **Snap By Area Factor** - Via groups are snapped by a maximum area equal to the **Tolerance** times the surface area of the via group being snapped.
    - **Snap By Distance** - Via groups are snapped by a maximum distance of the **Tolerance**.
2. Enter a value in the **Tolerance** field (e.g., **3**).
3. Check the **Remove Dangling Vias** box to delete vias that have no connectivity or have connectivity only to a single layer (checked by default).

Primitive Snapping

1. Enter a value in the **Tolerance** field, including the unit of measurement (e.g., **0.05um**).
2. Check the **Check Connectivity** box to ensure snapping does not violate the connectivity of the original objects (checked by default).

Via Grouping

1. Select a **Grouping Method** from the drop-down menu (**Group By Proximity** selected by default).
    - **Group By Density** - Uses hierarchical density-based spatial clustering to identify vias that form dense regions. This method provides superior detection of complex via patterns and automatically identifies clusters of varying shapes and sizes while handling noise and outliers.
    - **Group By Proximity** - Groups vias that are spaced similarly. Vias are not grouped if the distance between them exceeds the specified **Tolerance**.
    - **Group By Range** - Groups vias based on whether a potential member is separated by **no more** than the specified **Tolerance** from any other member.

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 value in the **Tolerance** field, including the unit of measurement (e.g., **100um**). This represents the gap distance (i.e., maximum inter-via distance for clustering).
3. Check the **Check Containment** box to detect and prevent electrical shorts (checked by default).

> **Note:**
>
> **Check Containment** analyzes the placement of via primitives. When the possibility of a short is detected, **Check Containment** splits the group to prevent a short.

Geometry Wrapping

Enter a value in the **Tolerance** field (e.g., **1um**).

Remove Small Metal Islands

Enter a value in the **Tolerance** field (e.g., **5 um^2**).

Reconstruct Arcs

Enter a value in the **Tolerance** field (e.g., **200nm**).

Remove Small Lines

Enter a value in the **Tolerance** field (e.g., **5e-06um**).

Remove Small Fragments

1. Check the **Remove Small Fragments** box to enable the feature.
2. Enter a value in the **Tolerance** field (e.g., **0um2**).

Snapping Vias in IC Mode

Via groups serve as an electrical connection between objects on different metal layers. Sometimes objects and via groups are misaligned by a small amount which can increase mesh complexity. Snapping via groups aligns the objects and via groups across layers.

Complete steps in the following subsections to configure **Via Snapping** settings, then snap (i.e., align) vias within a design.

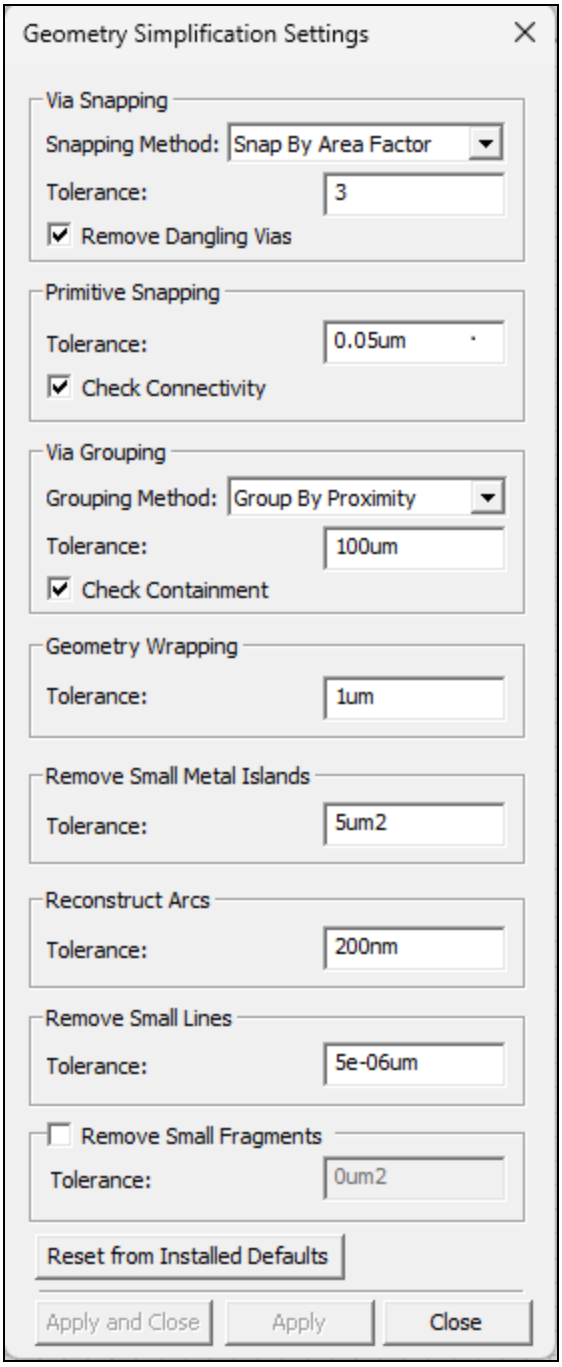
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Configuring Via Snapping Settings

1. From the **Layout** ribbon, click **Settings**.

The **Geometry Simplification Settings** 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.



2. From the **Via Snapping** area, do the following:

    a. Select a **Snapping Method** from the drop-down menu (**Snap By Area Factor** selected by default).

    - **Snap By Area Factor** - Via groups are snapped by a maximum area equal to the **Tolerance** times the surface area of the via group being snapped.

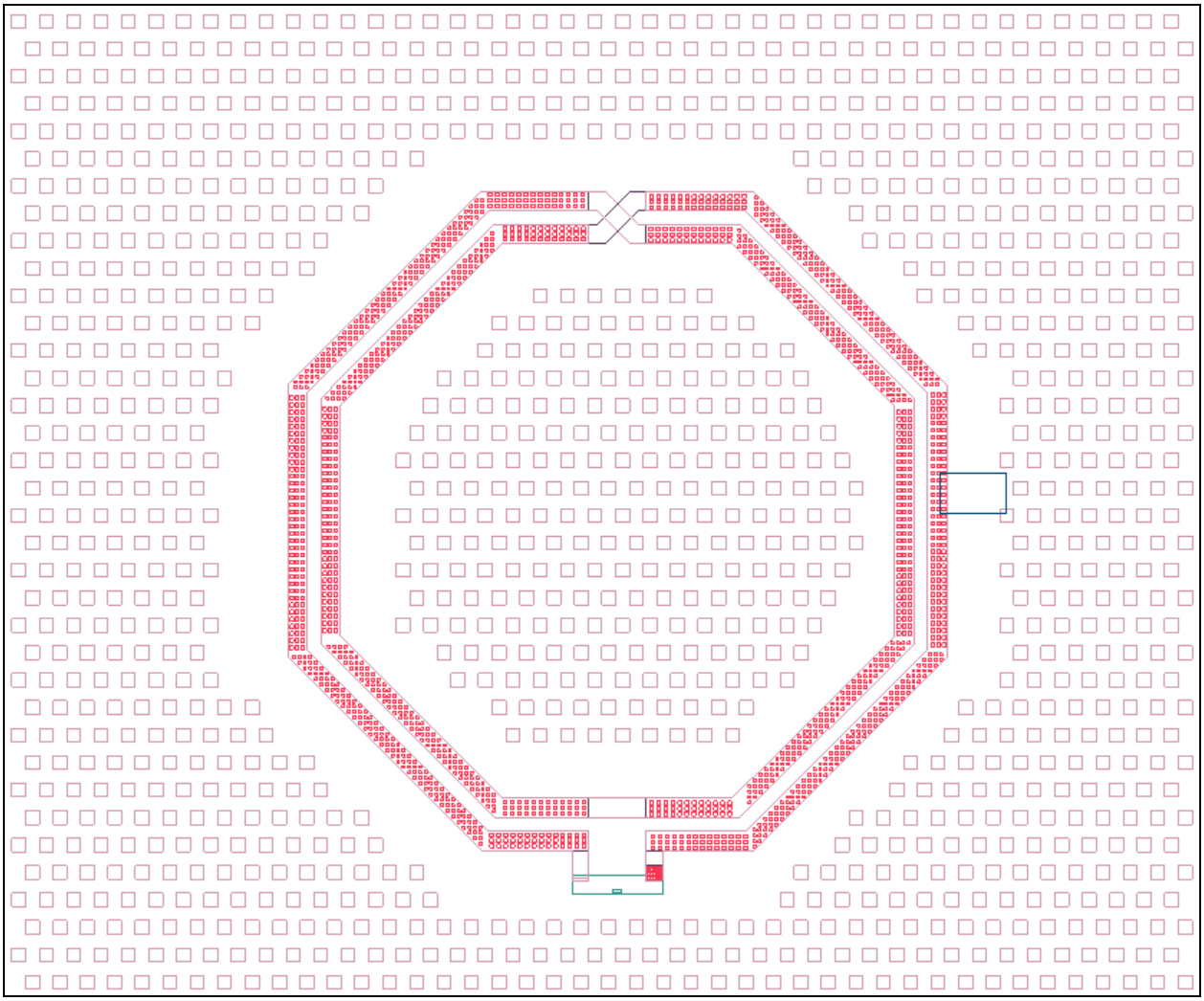
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Snap By Distance** - Via groups are snapped by a maximum distance of the **Tolerance**.

b. Enter a value in the **Tolerance** field (e.g., **3**).

c. Check the **Remove Dangling Vias** box to delete vias that have no connectivity or have connectivity only to a single layer (checked by default).

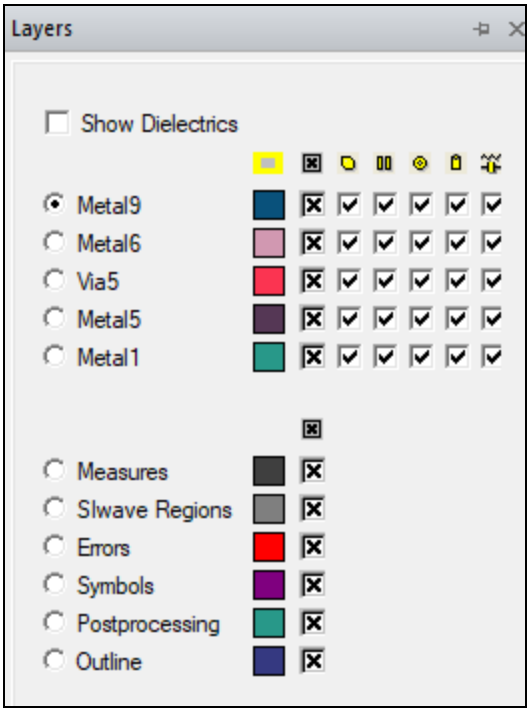
d. Click **Apply and Close**.



Snapping Vias on a Specific Layer

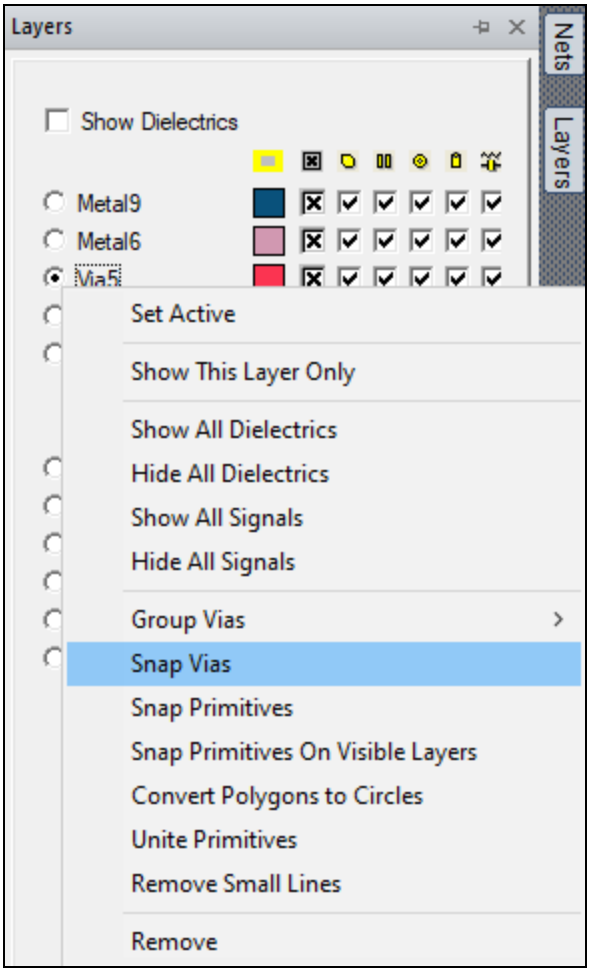
1. If the **Layers** window is not visible, navigate to the **Layers** tab. Click the **Layers** tab to open the window.

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



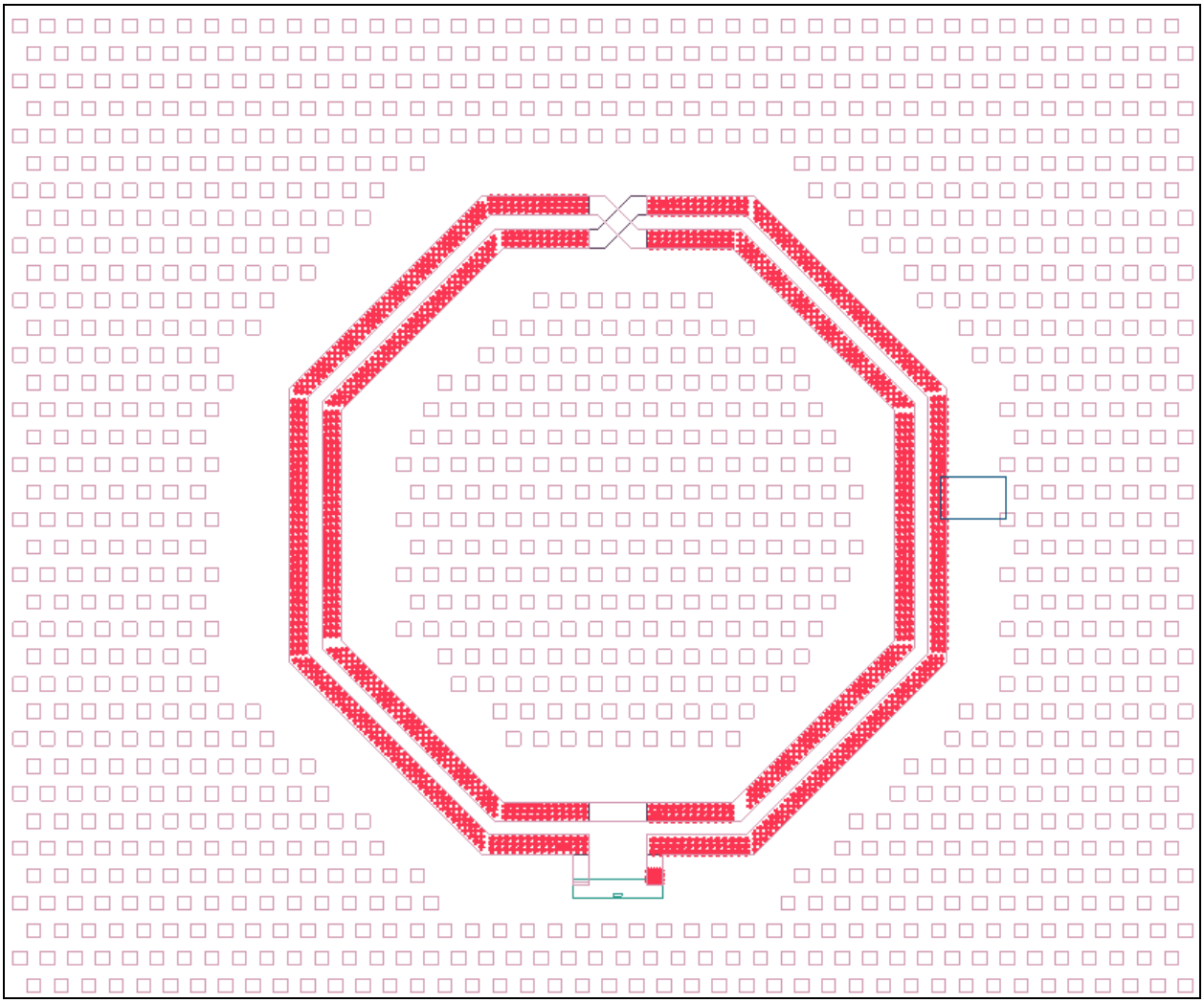
2. Right-click the selected layer (e.g., **Via5**) and select **Snap Vias**.

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



The vias in the design are aligned.

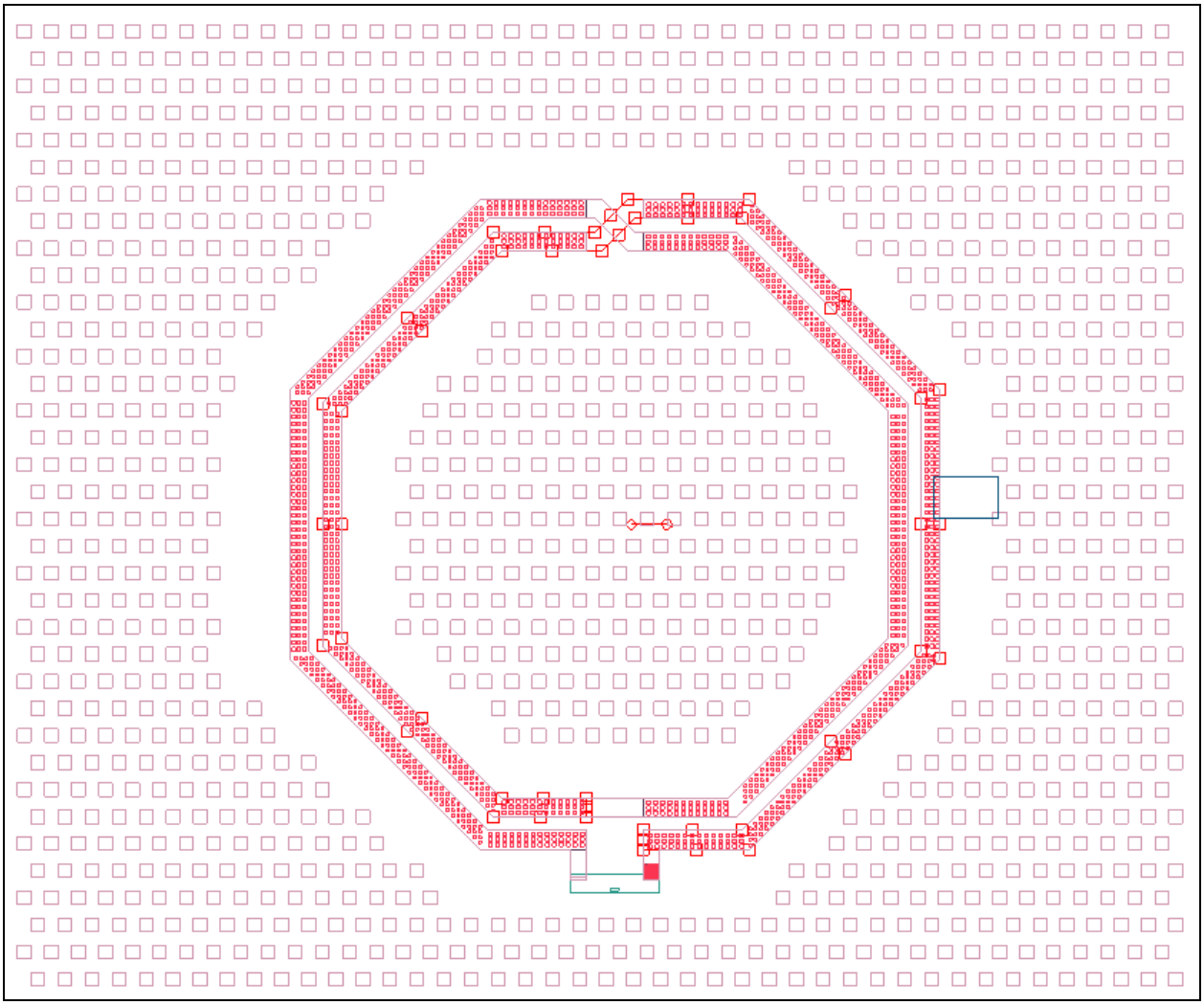
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

Snapping Vias in the Layout Editor

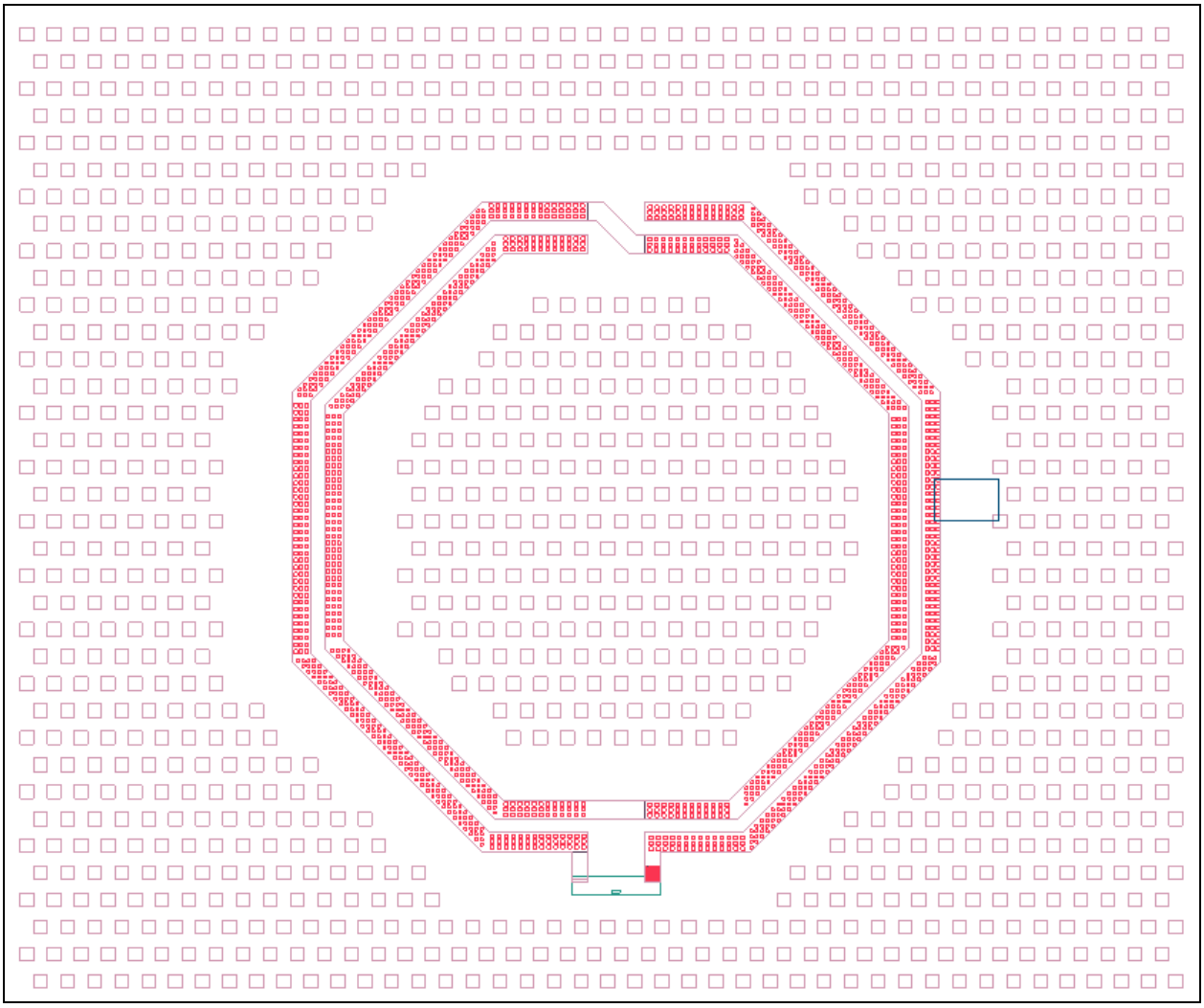
1. From the **Layout Editor**, select a group of via primitives.



> **Note:** Via primitives can be rectangles, polygons, or lines. Only rectangular via primitives can have more than one primitive in a via group. If you want to make a via group out of polygons or lines, the group can have only one line primitive or one polygon primitive in it.

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

2. From the **Layout** ribbon, click **Snap Vias** to align the selected vias.



Grouping Vias in IC Mode

Integrated chips are commonly fabricated using numerous small vias between signal layers which can result in highly complex geometries. Geometries can be simplified through grouping. Via grouping replaces via regions with an equivalent wrapping polygon that has material properties representative of the grouped primitives. The **Group Vias** tool detects clusters of primitives and creates via groups accordingly. Via groups can then be combined or dissolved depending on which option is selected.

Vias can be grouped using three methods: **Group By Density**, **Group By Proximity**, and **Group By Range**.

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

Group By Density

The density-based clustering method uses the Hierarchical Density-Based Spatial Clustering of Applications with Noise (HDBSCAN) algorithm to identify vias that form dense regions, providing superior detection of complex via patterns. It groups vias according to their density distribution across the layout, using the specified tolerance as the maximum inter-via distance for determining neighborhood relationships during density estimation. The method automatically identifies clusters of varying shapes and sizes, naturally handles noise and outliers by leaving low-density vias ungrouped, uses a hierarchy of density estimates to determine optimal cluster boundaries, and considers both local and global spatial relationships.

Group By Proximity

The proximity-based method groups vias that exhibit similar spacing patterns, forming clusters only when the distance between vias does not exceed the specified tolerance. It evaluates the relative spacing between nearby vias, identifies vias that share comparable inter-via distances, and produces clusters with consistent spacing characteristics. This method is well-suited for layouts where vias follow regular or repeating spacing patterns and where grouping should reflect those spacing relationships.

Group By Range

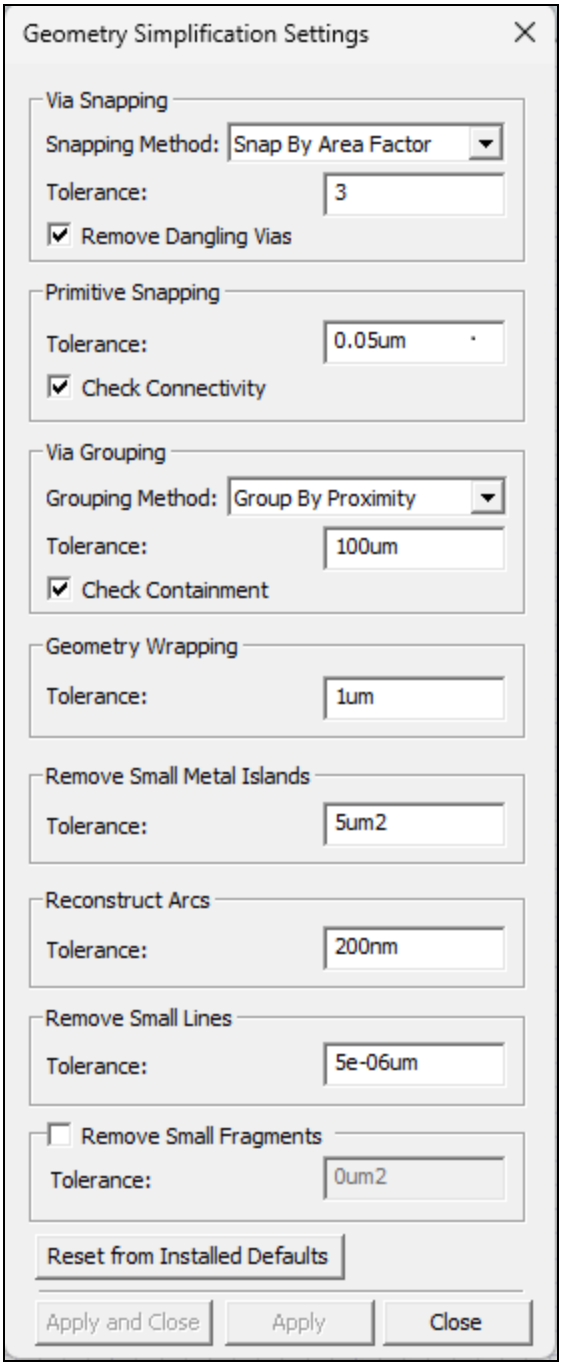
The range-based method groups vias by expanding outward from an initial seed via, adding new members whenever they fall within the specified tolerance of any existing cluster member. It grows clusters through transitive proximity relationships, allowing a via to join the group even if it is not close to the seed itself but is close to another member. This approach creates clusters that reflect chained proximity connections and is effective for capturing elongated or sequential via arrangements.

Configuring Via Grouping Settings

1. From the **Layout** ribbon, click **Settings**.

   The **Geometry Simplification Settings** 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.



2. Select a **Grouping Method** from the drop-down menu (**Group By Proximity** selected by default).

    - **Group By Density** – Uses density-based spatial clustering for superior detection of complex via patterns.

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

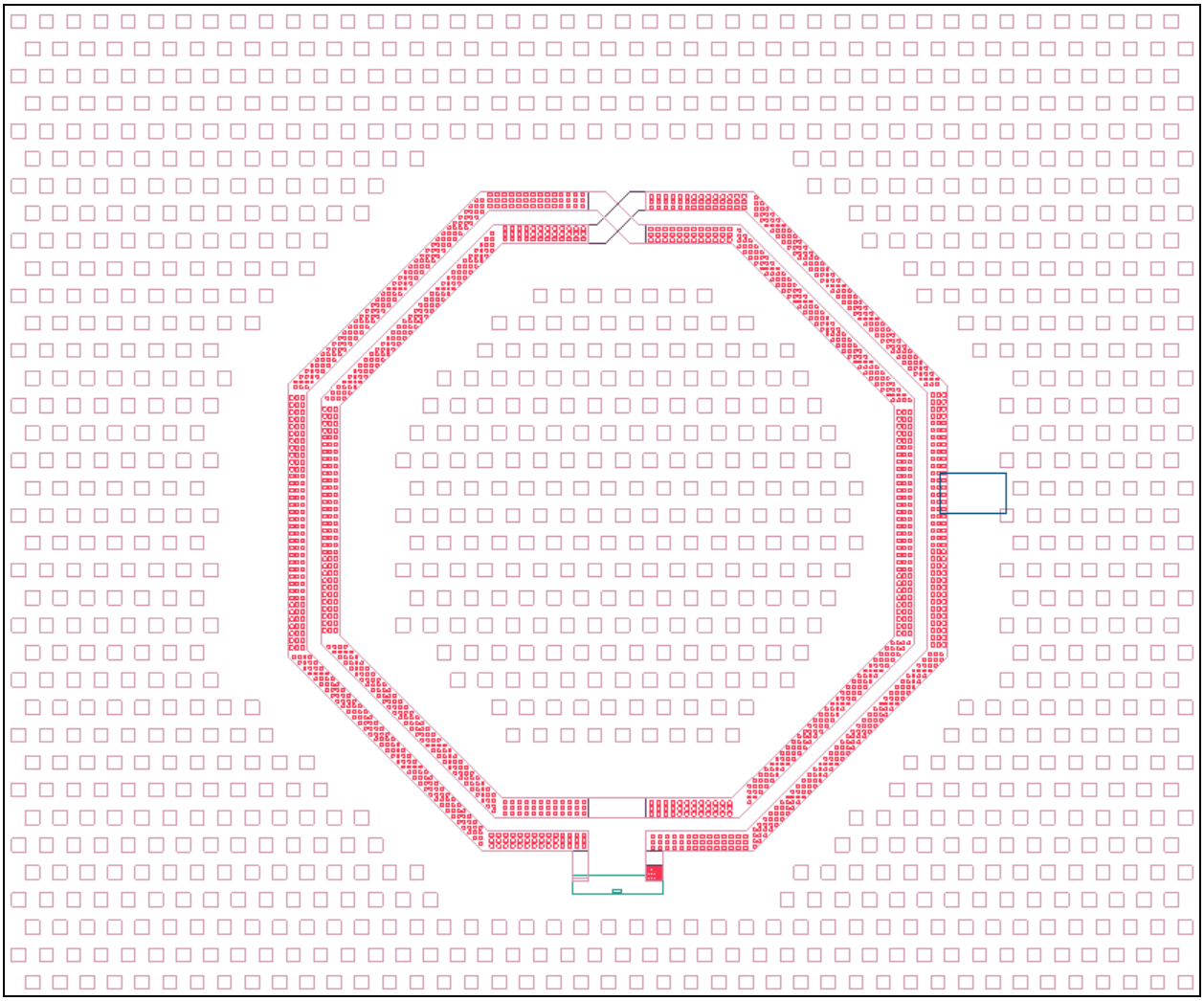
- **Group By Proximity** – Groups vias that are spaced similarly. Vias are not grouped if the distance between them exceeds the **Tolerance**.
- **Group By Range** – Groups vias based on whether a potential member is separated by no more than the **Tolerance** from any other member.

3. Enter a value in the **Tolerance** field, including the unit of measurement (e.g., 100um). This represents the gap distance or maximum inter-via distance for clustering.
4. Check the **Check Containment** box to detect and prevent electrical shorts (checked by default).

> **Note:**
>
> **Check Containment** analyzes the placement of via primitives. When the possibility of a short is detected, Check Containment attempts to prevent a short.

5. Click **Apply and Close**.

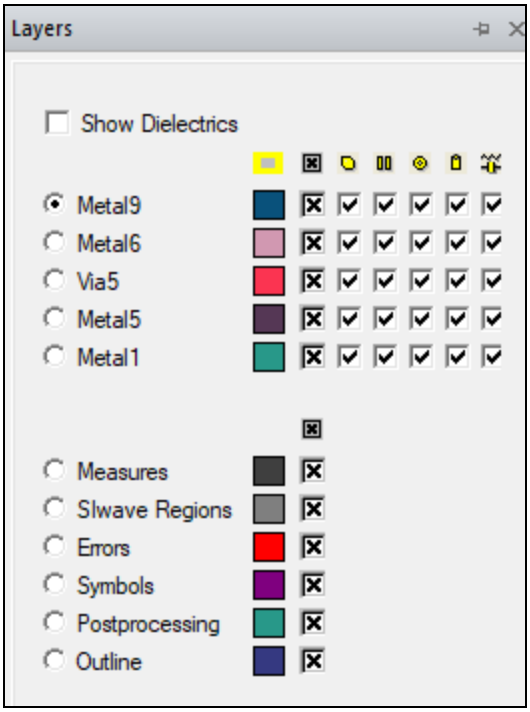
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Grouping Vias on a Specific Layer

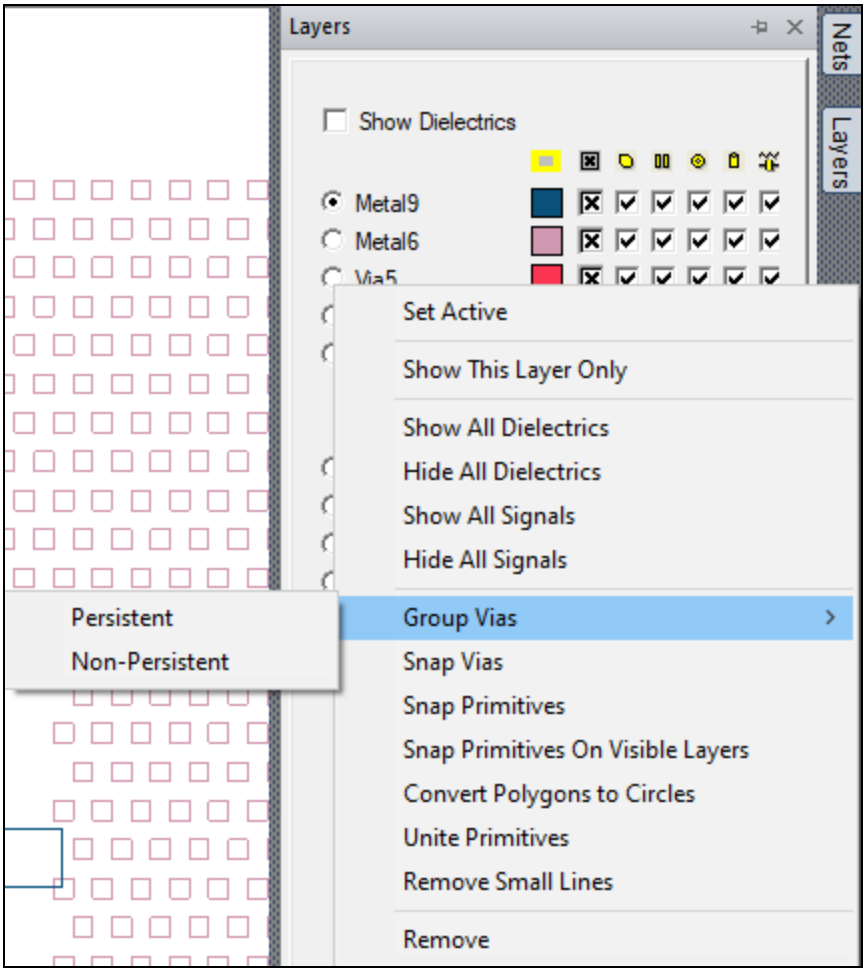
1. If the **Layers** window is not visible, navigate to the **Layers** tab. Click the **Layers** tab to open the window.

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



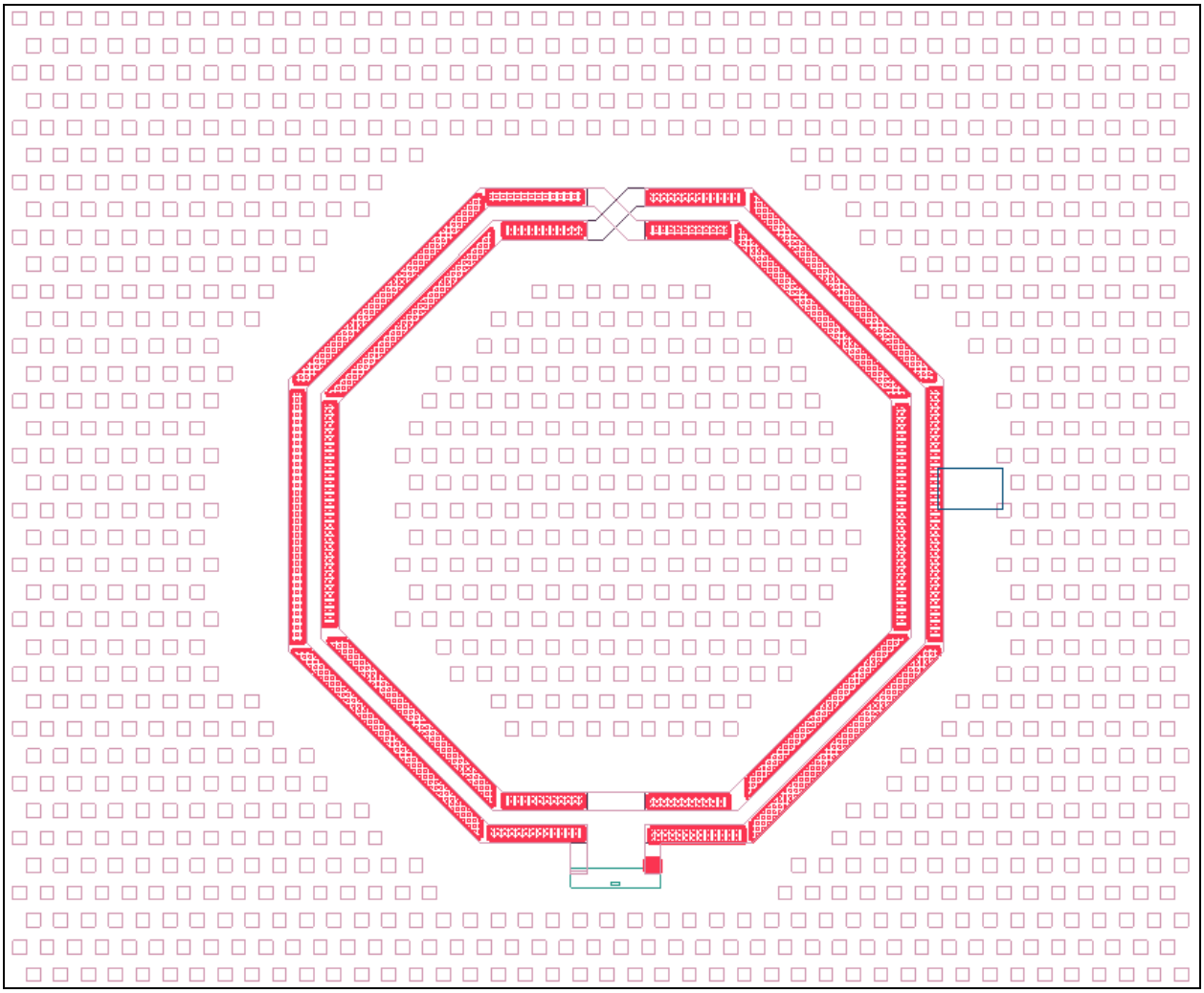
2. Right-click the selected layer (e.g., **Via5**) and select **Group Vias** > [***Option***]:
    - **Persistent** – Create via groups and keep original primitives.
    - **Non-Persistent** – Create via groups and delete original primitives.

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



The vias in the design are grouped according to the selected grouping method and specified tolerance.

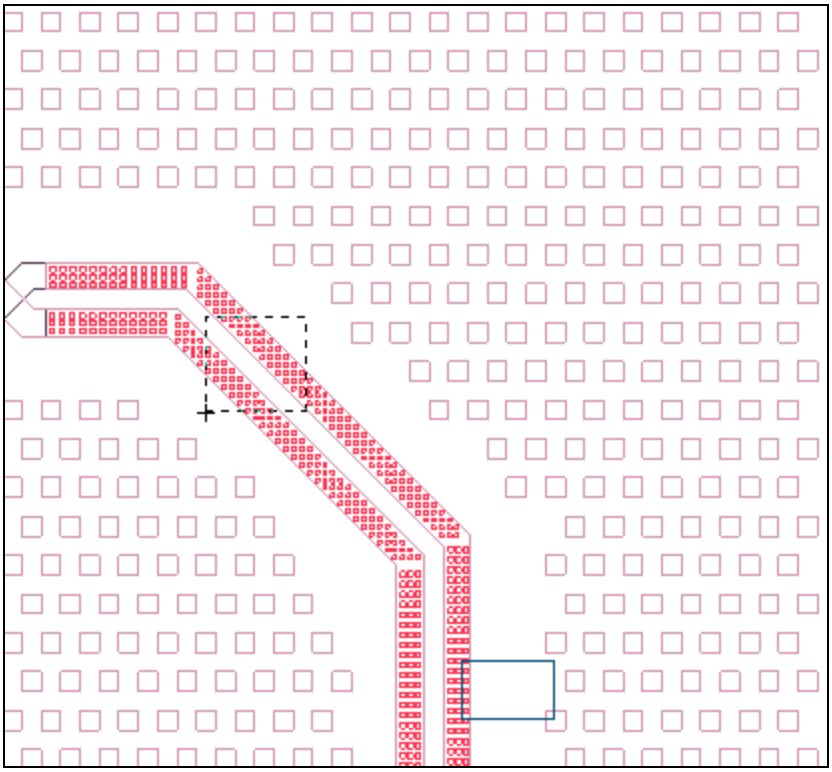
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

Manually Grouping Vias in the Layout Editor

1. Select a group of via primitives.

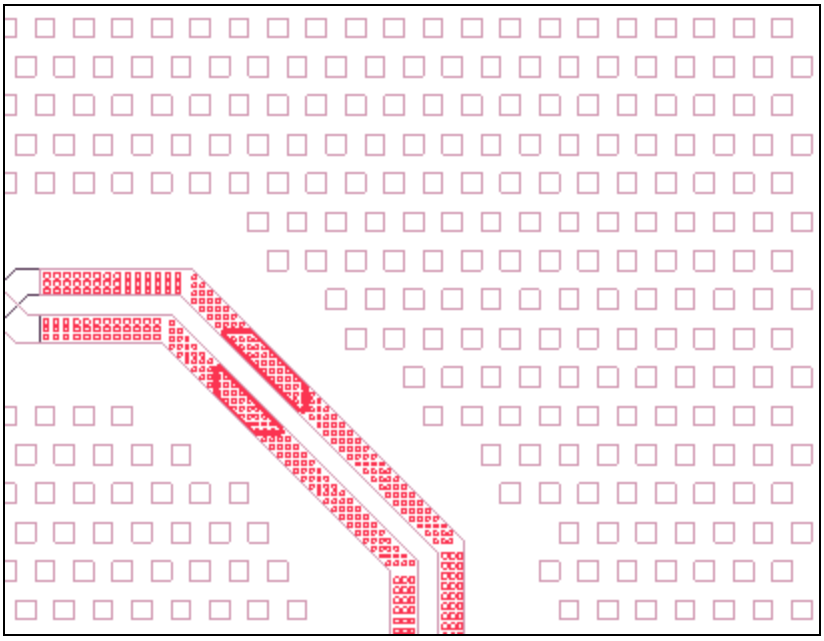


> **Note:**
>
> Via primitives can be rectangles, polygons, or lines. Only rectangular via primitives can have more than one primitive in a via group. To make a via group out of polygons or lines, the group can have only one line primitive or one polygon primitive in it.

2. From the **Layout** ribbon, select **Group Vias** > **[*Option*]**.
   - **Persistent** – Create a via group and keep the original primitives in the design
   - **Non-Persistent** – Create a via group and delete the original grouped primitives
   - **Combine** – Merge selected via groups and/or via primitives into a new via group

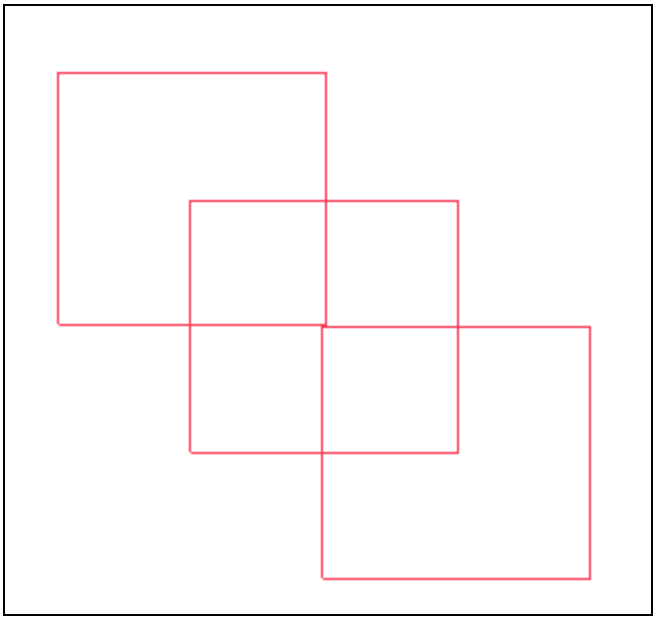
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Dissolve** – Restore the selected via group into the primitives that comprise it



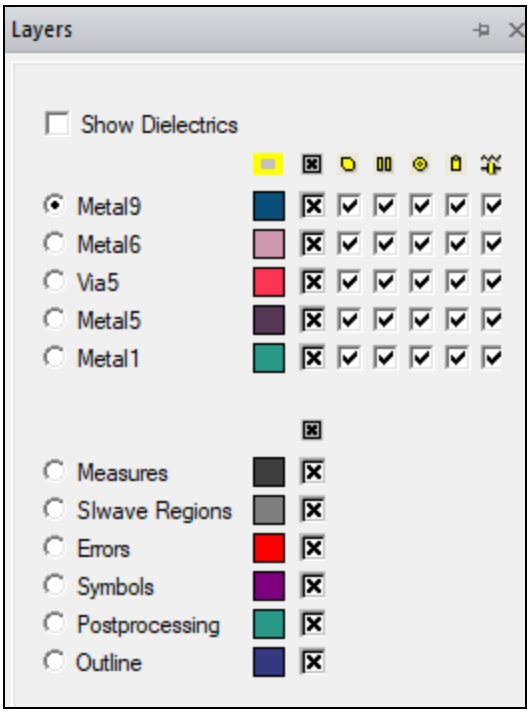
Uniting Primitives in IC Mode

Follow these steps to unite two or more primitives that overlap on the same layer.



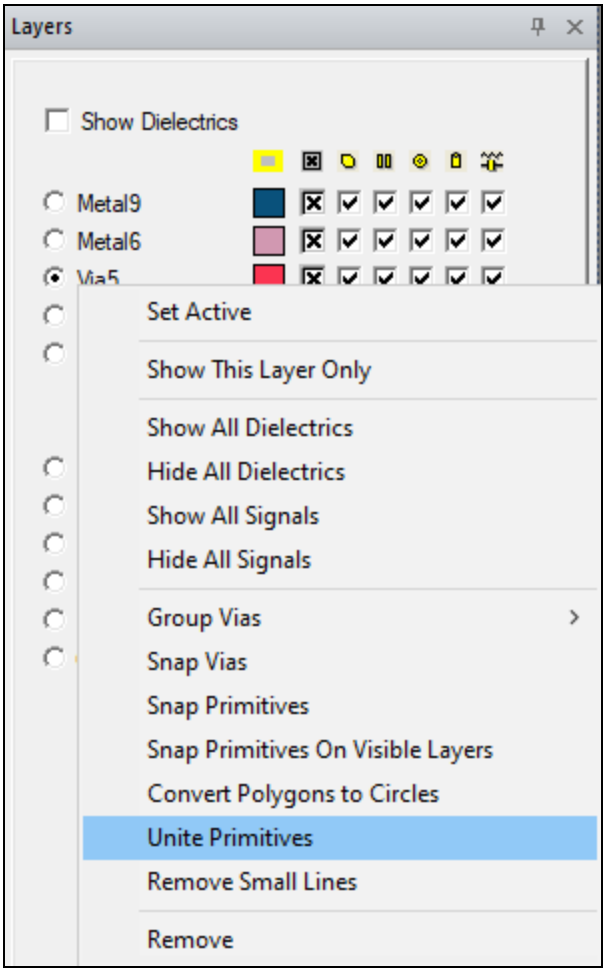
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. If the **Layers** window is not visible, navigate to the **Layers** tab. Click the **Layers** tab to open the window.



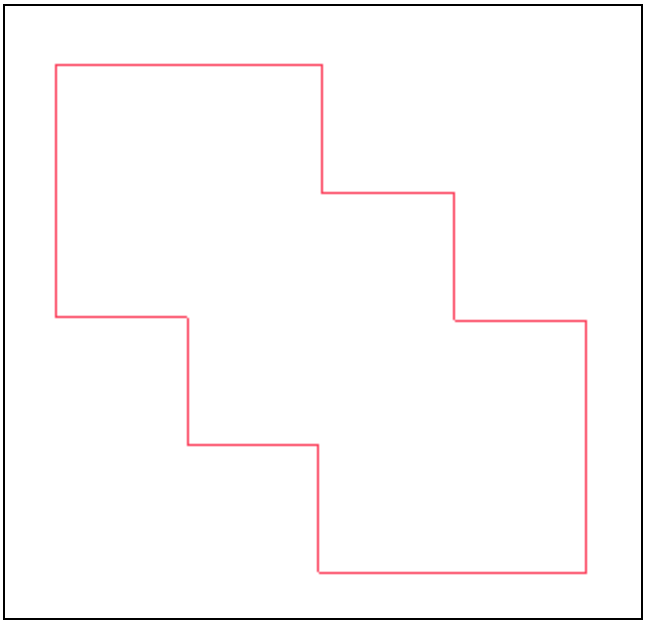
2. Right-click the selected layer (e.g., **Via5**) and select **Unite Primitives**.

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



Any overlapping primitives merge and their internal boundaries (i.e., the overlapping aspects of the primitives) are removed.

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



Reconstructing Arcs in IC Mode

Imported designs may contain arcs that have been approximated as many small line segments. These segmented arcs increase geometric complexity and can negatively affect simulation performance. Users can clean up a design by using the **Reconstruct Arcs** feature. Arcs are reconstructed when the segmented geometry meets the user-specified **Tolerance** threshold.

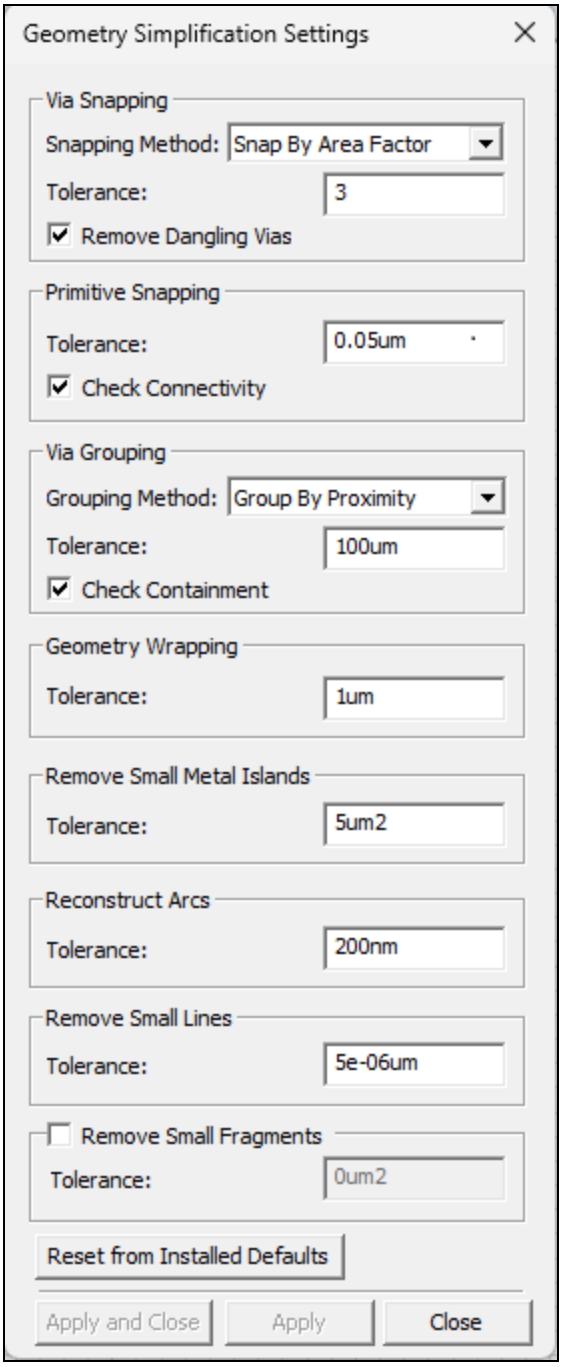
Complete steps in the following subsections to configure **Reconstruct Arcs**, then reconstruct arcs within a design.

Configuring Reconstruct Arcs Settings

1. From the **Layout** ribbon, click **Settings**.

   The **Geometry Simplification Settings** 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.



2. From the **Reconstruct Arcs** area, enter a value in the **Tolerance** field (e.g., **0.5um**). This value represents the minimum segmentation threshold at which an arc will be reconstructed.
3. 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.

Reconstructing Arcs in the Layout Editor

1. Select the appropriate elements of the design.
2. Do one of the following:

- From the **Layout** ribbon, click **Reconstruct Arcs**.
- Right-click within the **Layout Editor** and select **Reconstruct Arcs**.

The selected segmented arcs are compared to the value in the **Tolerance** field. Any arc whose segmentation falls within the set **Tolerance** is reconstructed into a smooth arc.

For other instructions on the usage of the **Layout Editor**, keyboard commands, and simple tasks like converting polygons to circles, refer to Getting Started with Ansys Electronics Desktop.

Wrapping Geometry in IC Mode

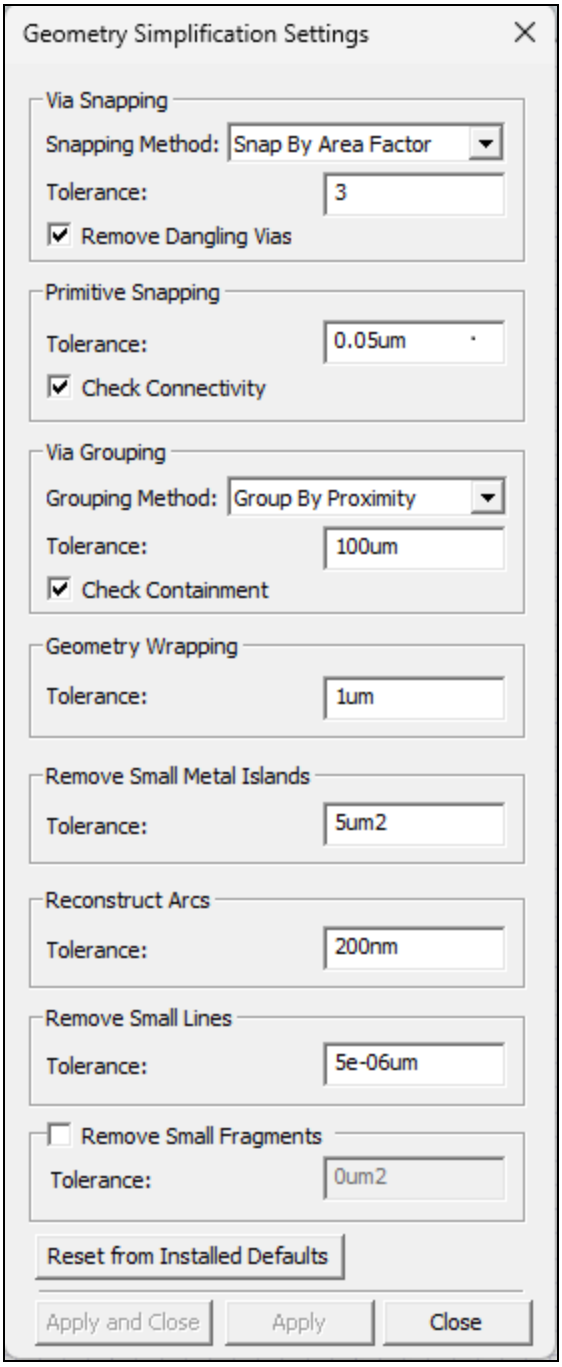
The **Wrap Geometry** feature greatly reduces mesh complexity by replacing groups of primitives with an equivalent wrapping polygon.

Selected primitives of or within the user-specified **Tolerance** distance can be wrapped by completing these steps.

1. From the **Layout** ribbon, click **Settings**.

The **Geometry Simplification Settings** window opens.

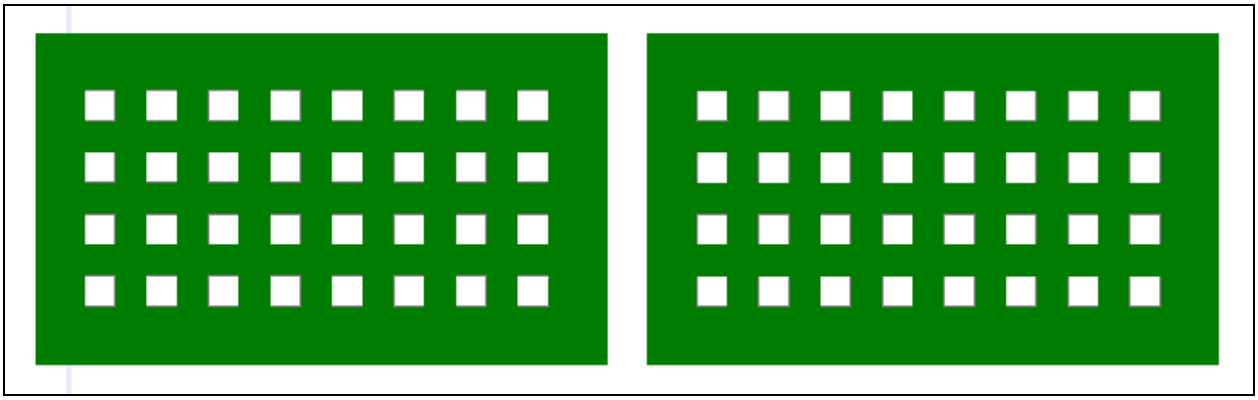
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



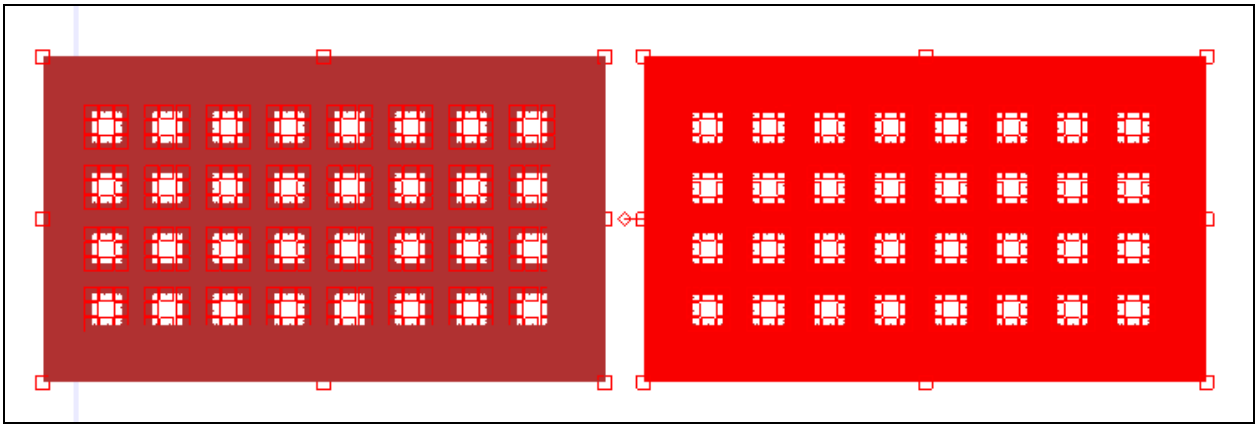
2. From the **Geometry Wrapping** area, enter a value in the **Tolerance** field (e.g., **1um**).
3. Click **Apply and Close**.

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

4. If applicable, navigate to the **Layout Editor** tab and the selected design geometry.



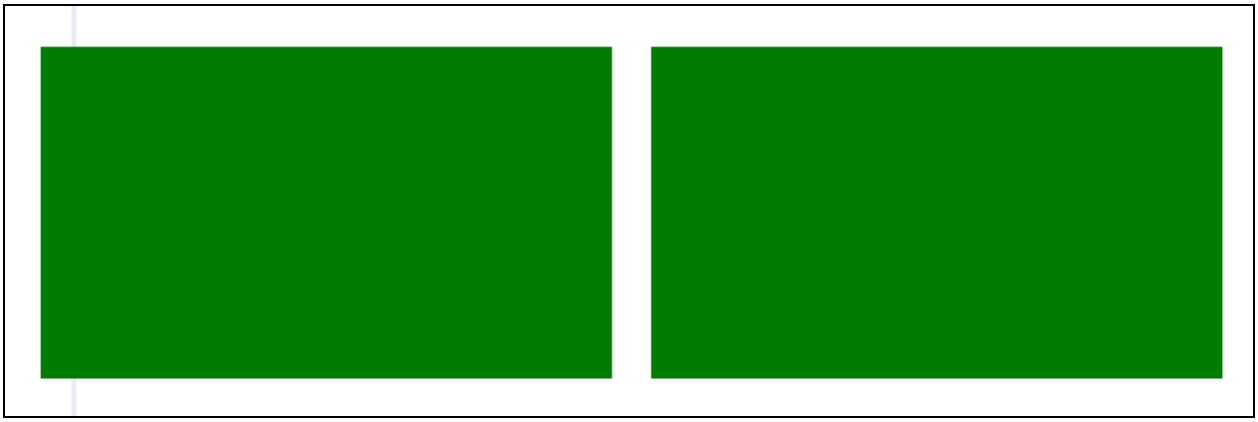
5. From the **Layout Editor**, select the appropriate geometry.



6. From the **Layout** ribbon, click **Wrap Geometry** and the selected geometry is compared to the value in the **Tolerance** field. Every element that meets or falls within the value is

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

wrapped



Snapping Primitives in IC Mode

Sometimes primitives exist directly above or below each other on adjacent layers, or they are misaligned by a small amount which can increase mesh complexity. Snapping primitives aligns these objects.

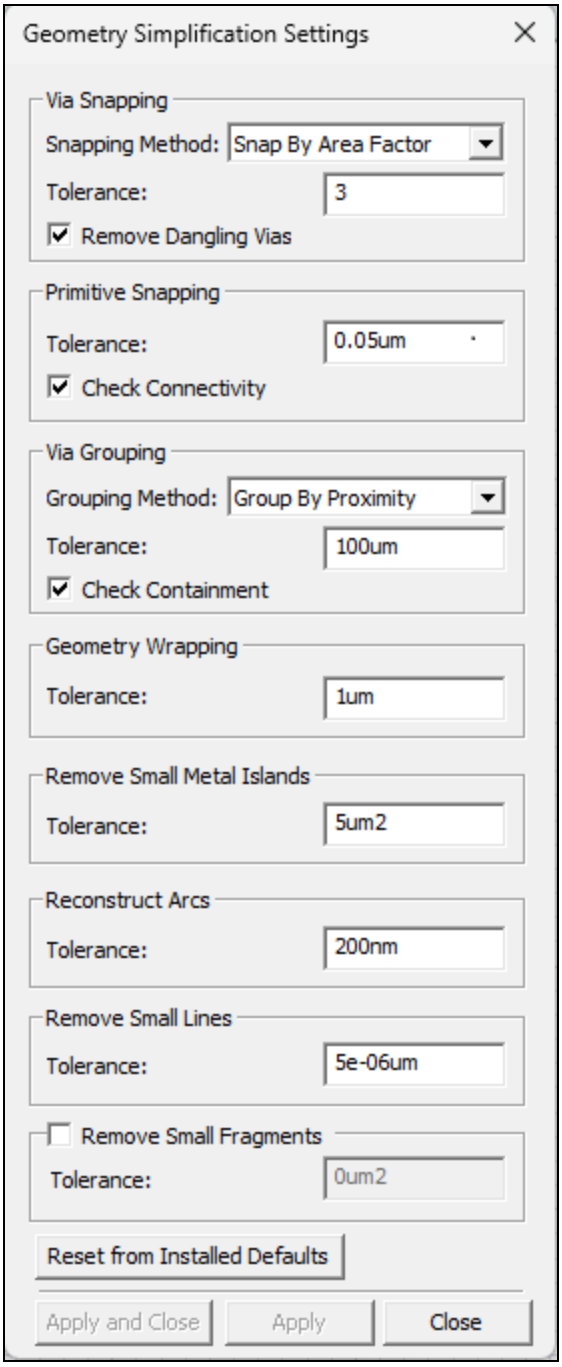
Complete steps in the following subsections to configure **Primitive Snapping**, then snap primitives in several ways.

Configuring Primitive Snapping Settings

1. From the **Layout** ribbon, click **Settings**.

The **Geometry Simplification Settings** 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.



2. From the **Primitive Snapping** area, do the following:

   a. Enter a value in the **Tolerance** field, including the unit of measurement (e.g., **1um**, although the default value is **0.05um**).

   b. Check the **Check Connectivity** box to ensure snapping does not violate the

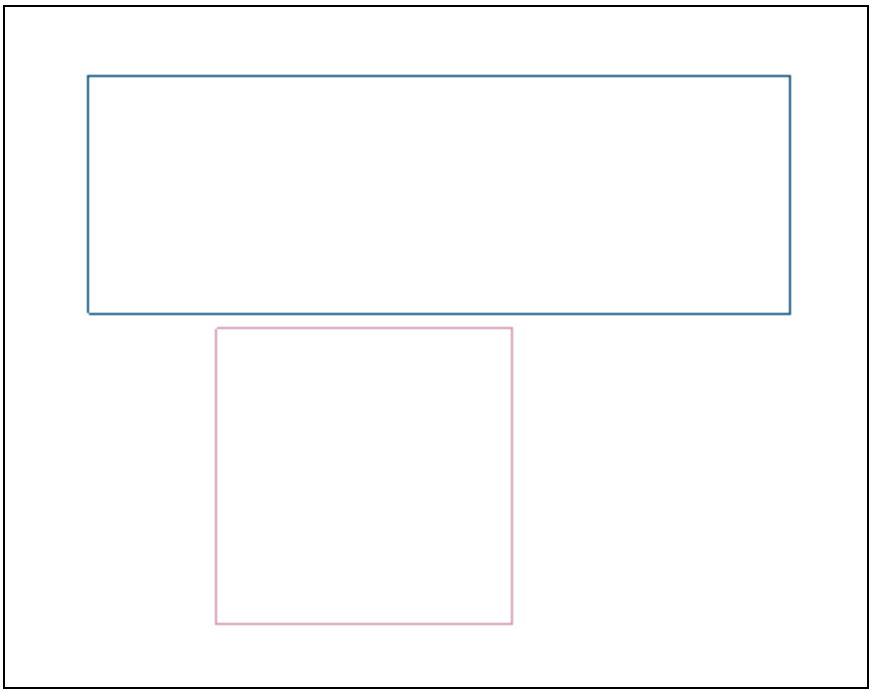
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

connectivity of the original objects (checked by default).

c. Click **Apply and Close**.

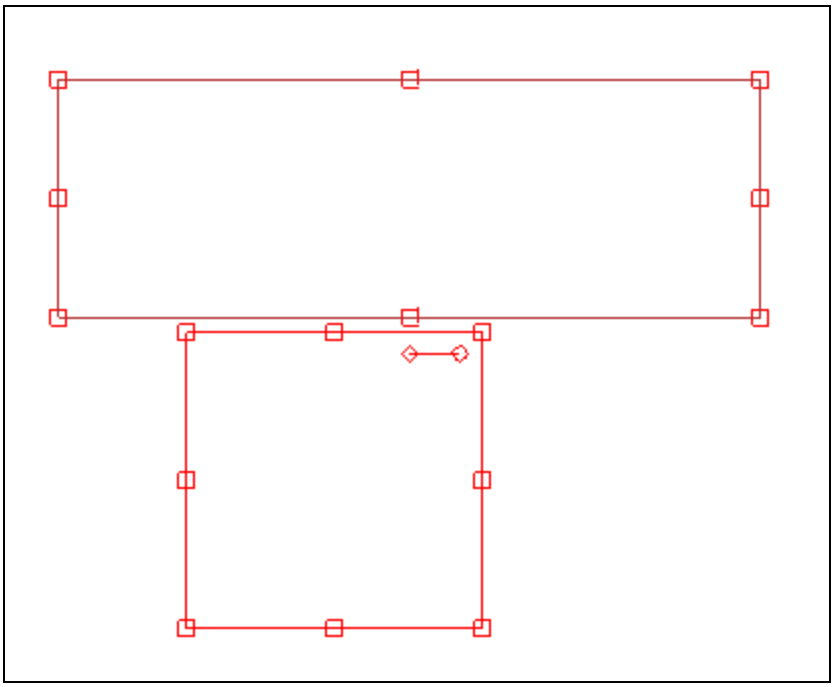
Snapping Primitives in the Layout Editor

1. If applicable, navigate to the **Layout Editor** tab and the selected components.



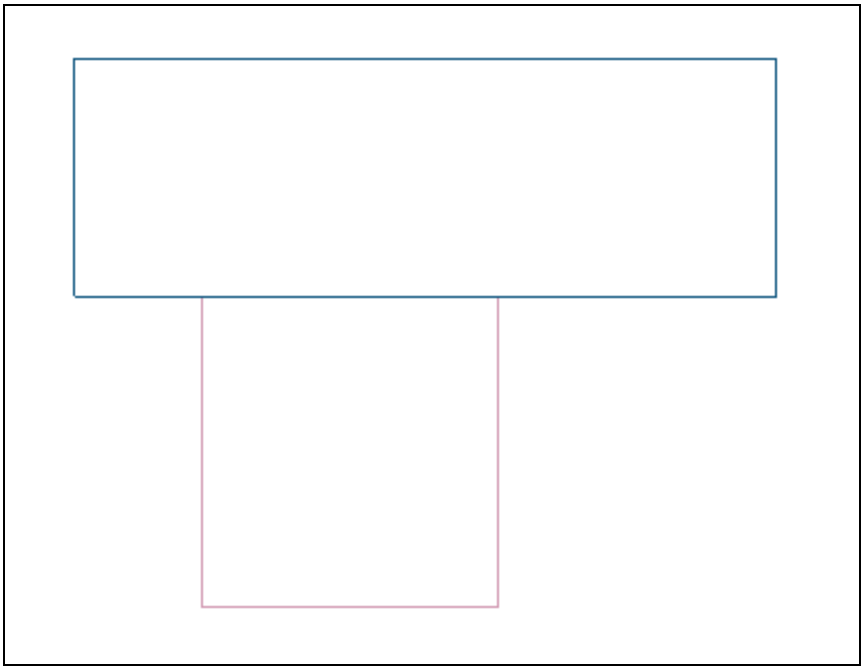
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the appropriate primitives.



3. From the **Layout** ribbon, click **Snap Primitives** and the selected primitives are compared to the value in the **Tolerance** field. Any primitives at or within the selected **Tolerance** are snapped (i.e., aligned).

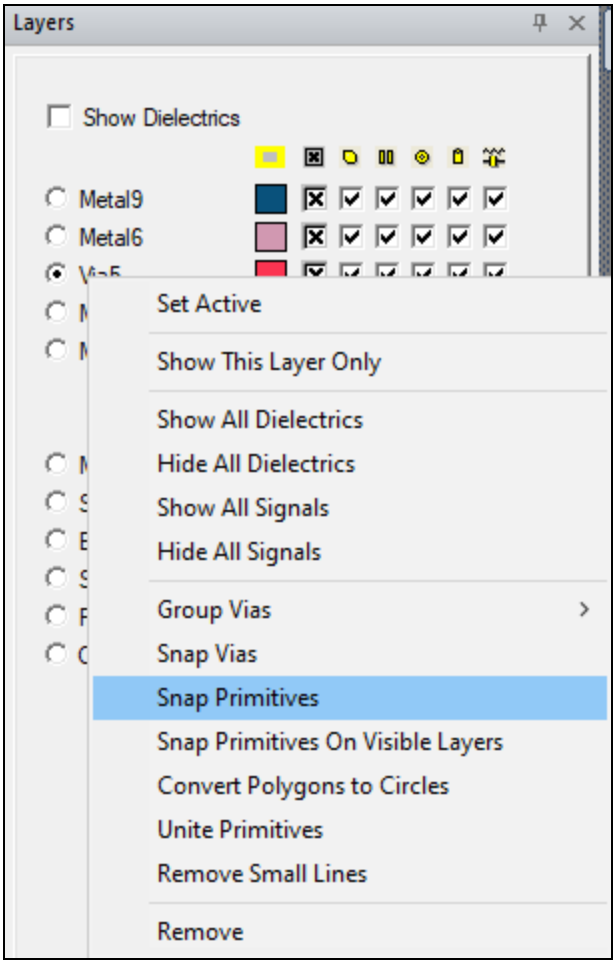
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Snapping Primitives on a Specific Layer

From the **Layers** window, right-click on a layer in the list (e.g., **Via5**) and select **Snap Primitives**.

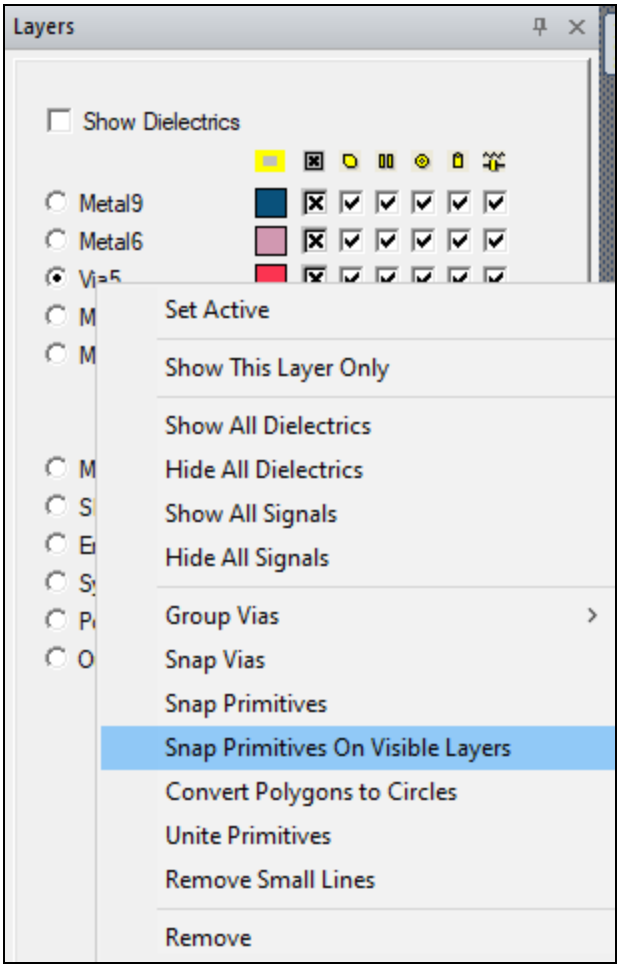
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Snapping Primitives On Visible Layers

From the **Layers** window, right-click on a layer in the list (e.g., **Via5**) and select **Snap Primitives On Visible Layers**.

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



Removing Objects and Layers in IC Mode

IC mode offers several ways to remove unwanted objects in a design (e.g., removing islands):

- Removing Small Islands in IC Mode
- Removing Small Lines in IC Mode
- Removing Small Holes in IC Mode
- Cropping and Removing Small Fragments in IC Mode

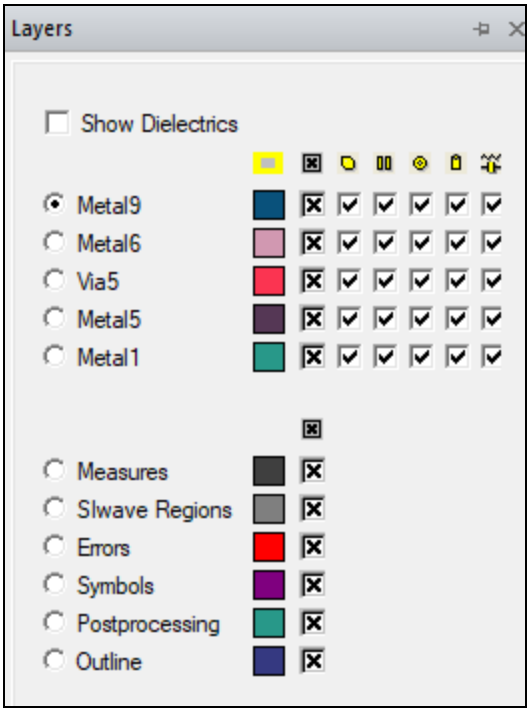
Follow these steps to remove objects from a layer.

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

Note:

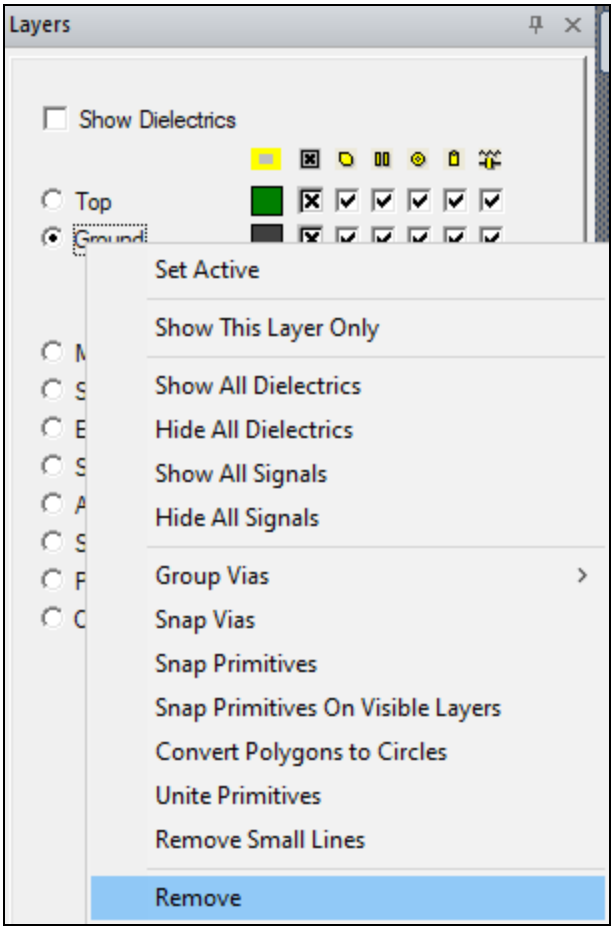
To remove layers from an encrypted design, refer to Trimming Layers from an Encrypted Design.

1. If the **Layers** window is not visible, navigate to the **Layers** tab. Click the tab to open the window.

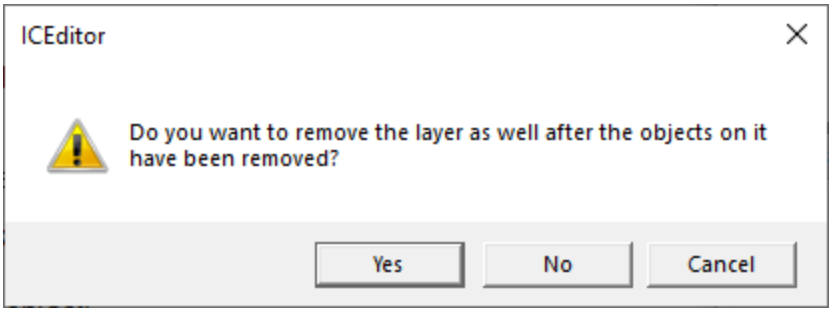


2. Right-click the appropriate layer (e.g., **Ground**) and select **Remove** to remove all the objects from it.

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 **Yes** or **No**, as appropriate.



Trimming Layers from an Encrypted Design

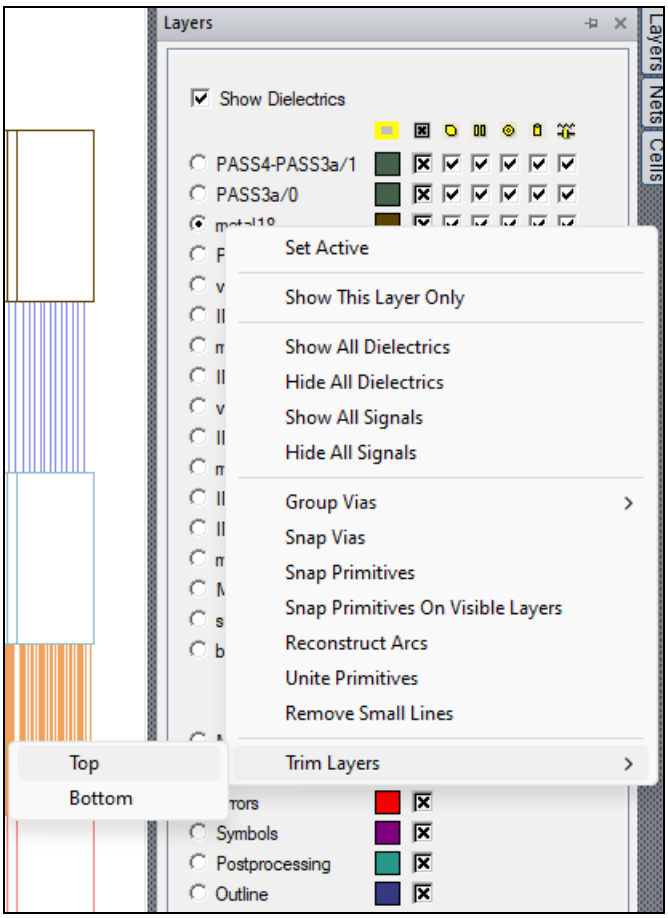
To enable interfacing with encrypted technology in hierarchical designs, users can remove the top and/or bottom layers to access the internal components. Follow these steps to remove the top/bottom layer from the design.

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

Note:

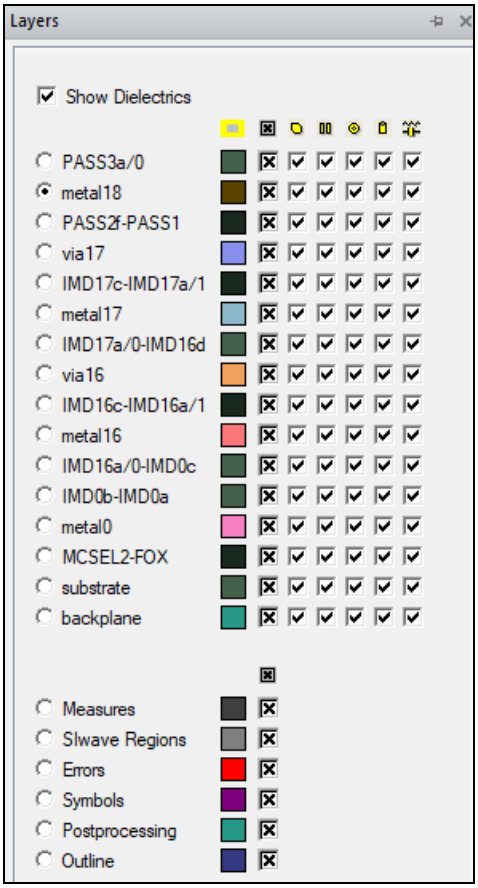
Trimming the top/bottom layers from a design disassociates the encrypted technology and invalidates all solutions and Green's Function Data.

1. Right-click in the **Layers** tab and select **Trim Layers** > **Top** or **Bottom**.



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 top or bottom layer is removed from the design.



Removing Small Islands in IC Mode

Designs may contain small detached pieces of metal (i.e., islands) that are unimportant to the simulation. An island is made up of one or more primitives and vias. Users can clean up a design by using the **Remove Islands** feature. Islands are removed if the total area covered by all of its primitives and vias is smaller than the user-specified **Tolerance**.

Complete steps in the following subsections to configure **Remove Small Metal Islands**, then remove islands within a design.

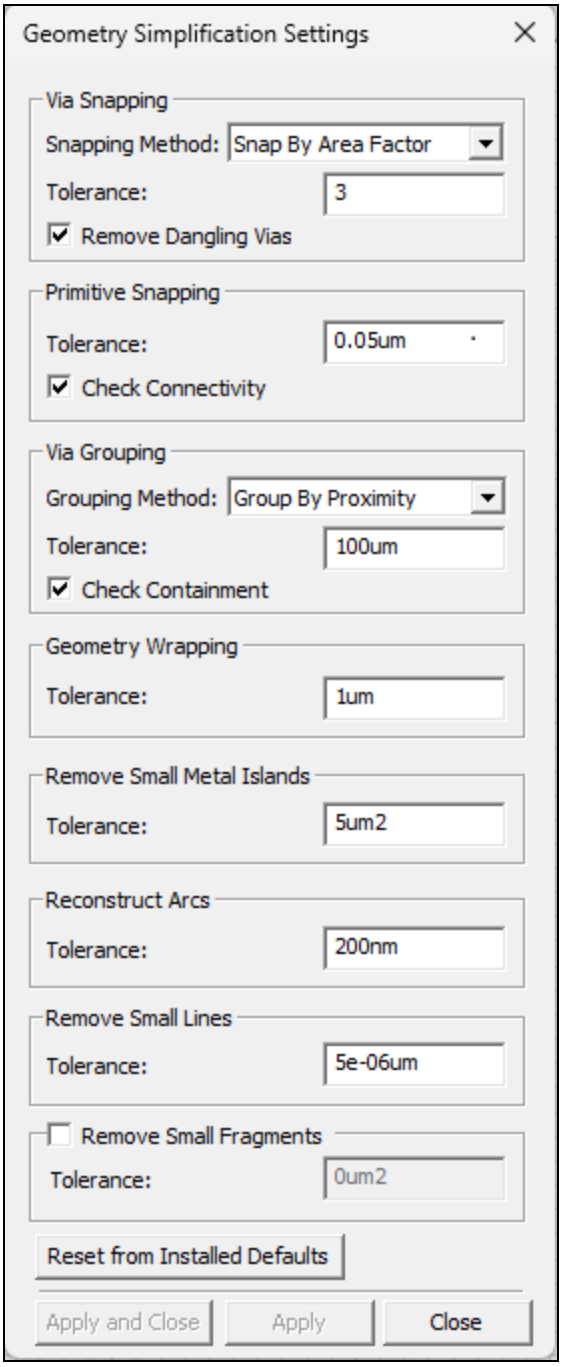
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Configuring Remove Small Islands Settings

1. From the **Layout** ribbon, click **Settings**.

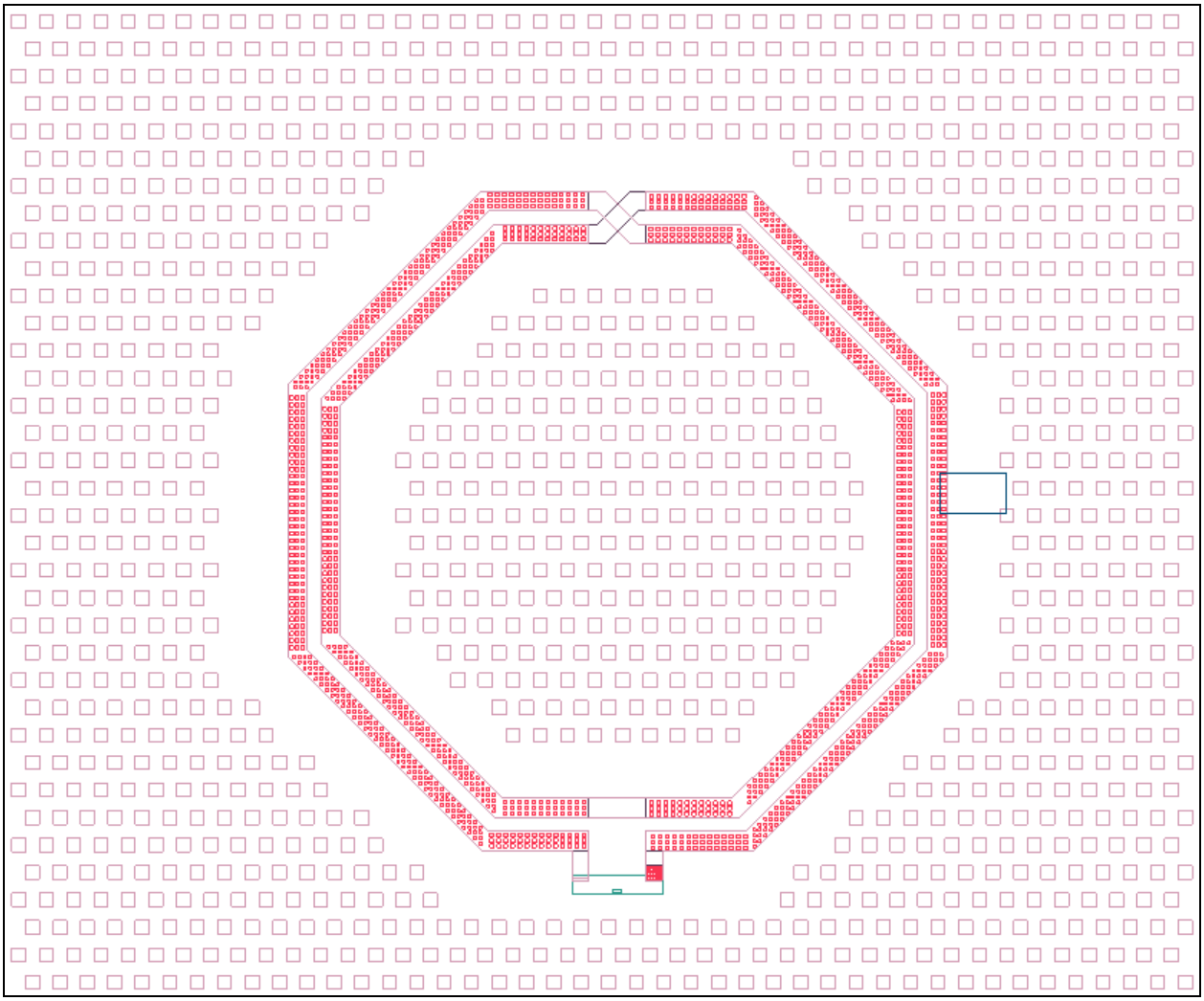
   The **Geometry Simplification Settings** window opens.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Remove Small Metal Islands** area, enter a value in the **Tolerance** field (e.g., **5um2**).
3. Click **Apply and Close**.

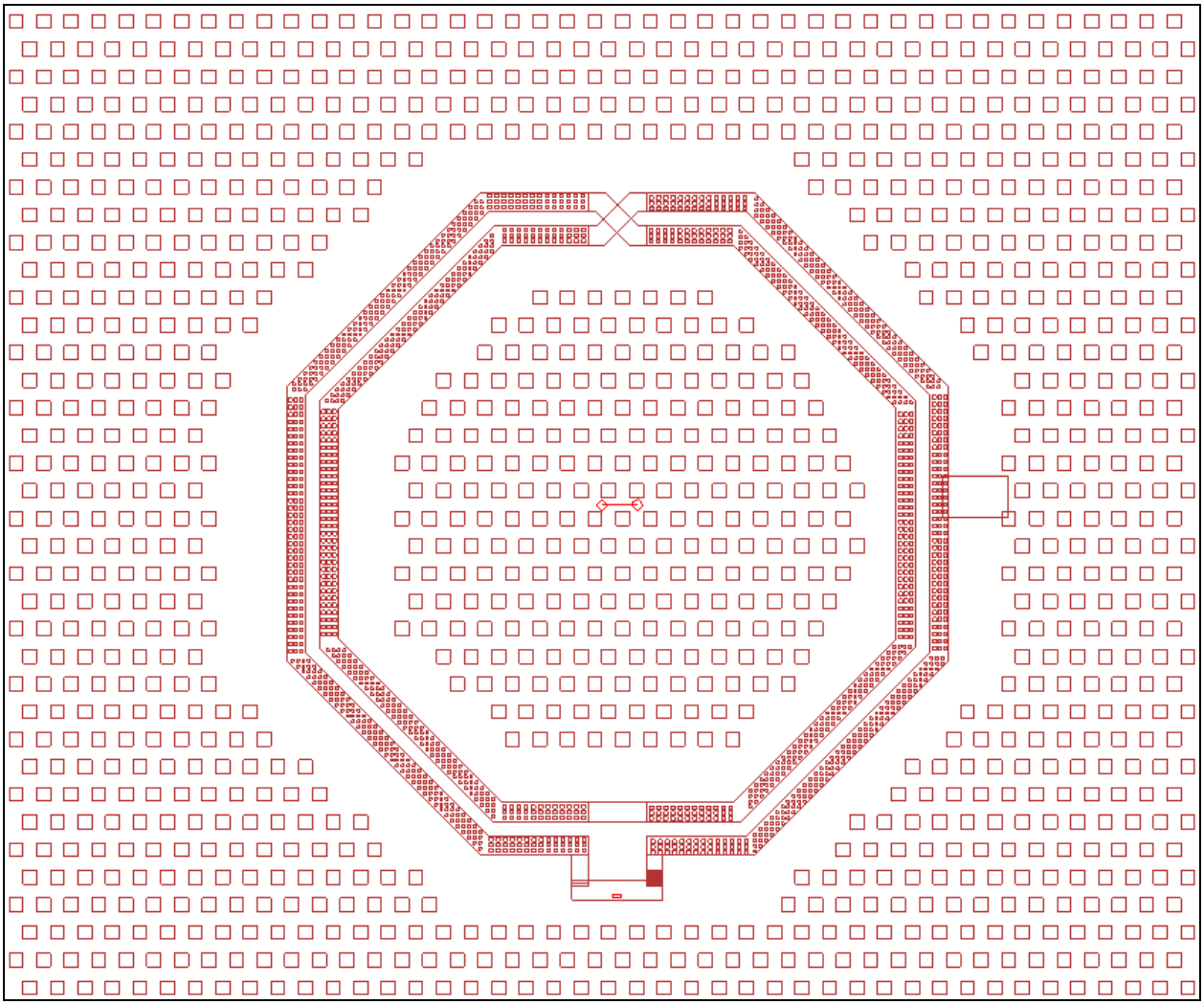
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Removing Small Islands in the Layout Editor

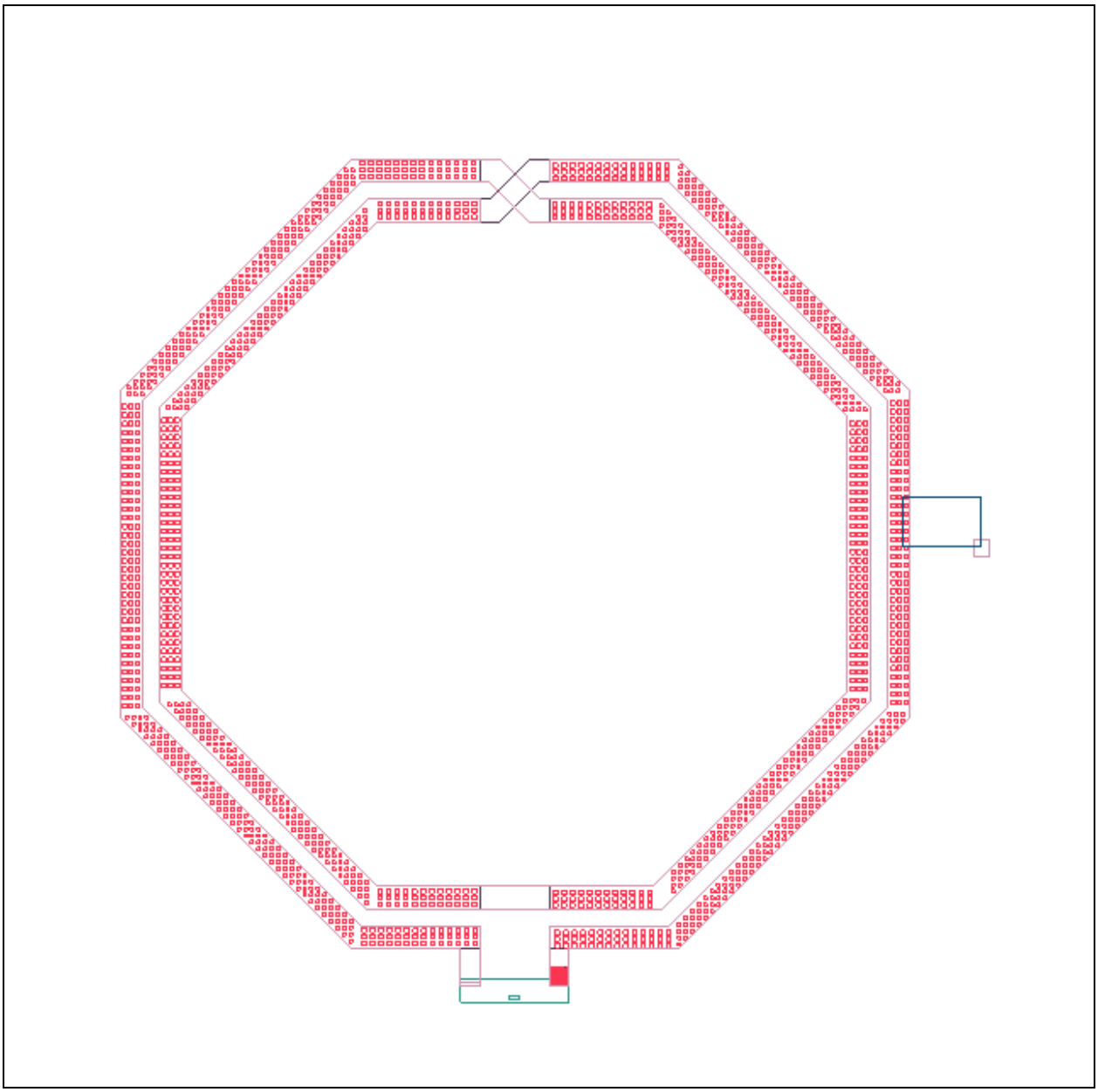
1. Select the appropriate elements of the design.



2. From the **Layout** ribbon, click **Remove Islands** and the selected elements are compared to the value in the **Tolerance** field. Any piece of metal the same size or smaller than the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

set **Tolerance** is removed from the design.



Removing Small Lines in IC Mode

Designs contain small lines (i.e., traces) that are unimportant to the simulation. Users can clean up a design by using the **Remove Small Lines** feature. Lines are removed if their total area is smaller than the user-specified **Tolerance**. Follow these steps to configure and run the **Remove Small Lines** feature.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

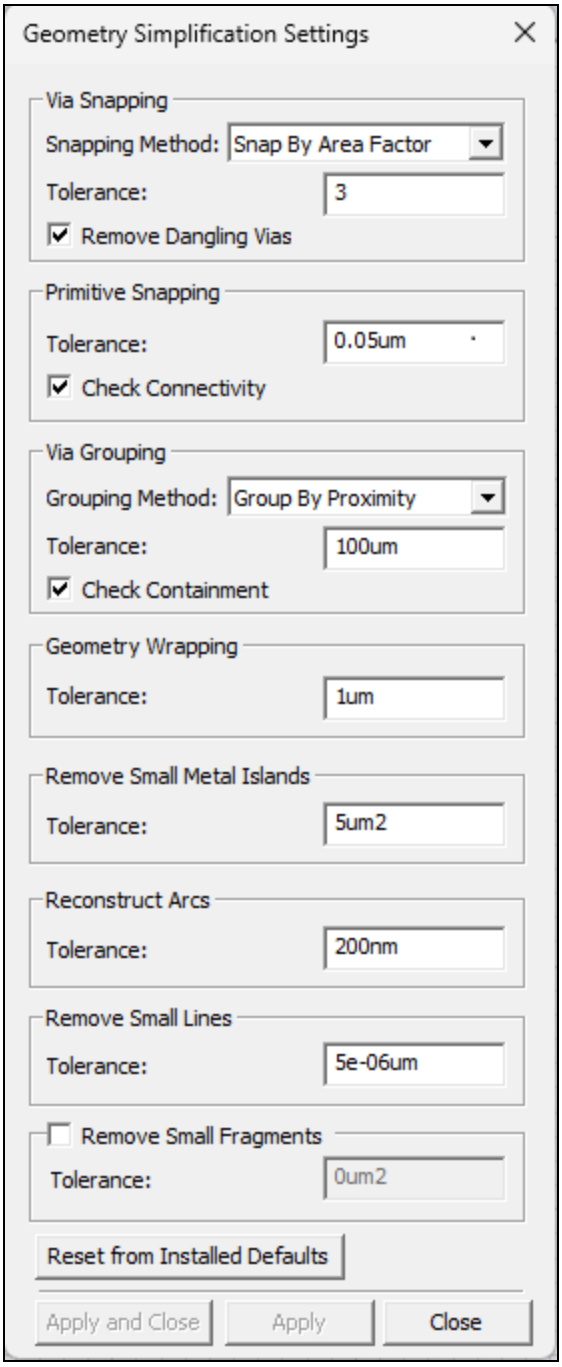
Complete steps in the following subsections to configure **Remove Small Lines**, then remove lines within a design.

Configuring Remove Small Lines Settings

1. From the **Layout** tab, click **Settings**.

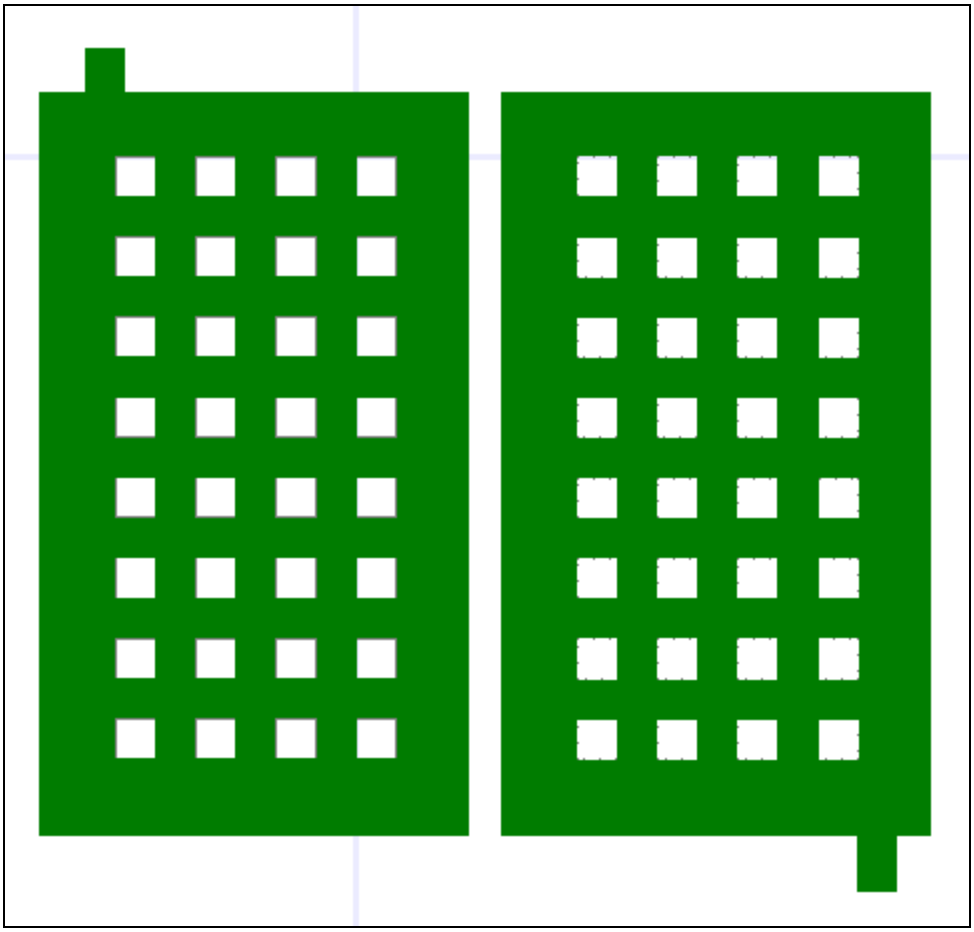
   The **Geometry Simplification Settings** window opens.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Remove Small Lines** area, enter a value in the **Tolerance** field (e.g., **1e-05**, although the default value is **5e-06um**).
3. Click **Apply and Close**.

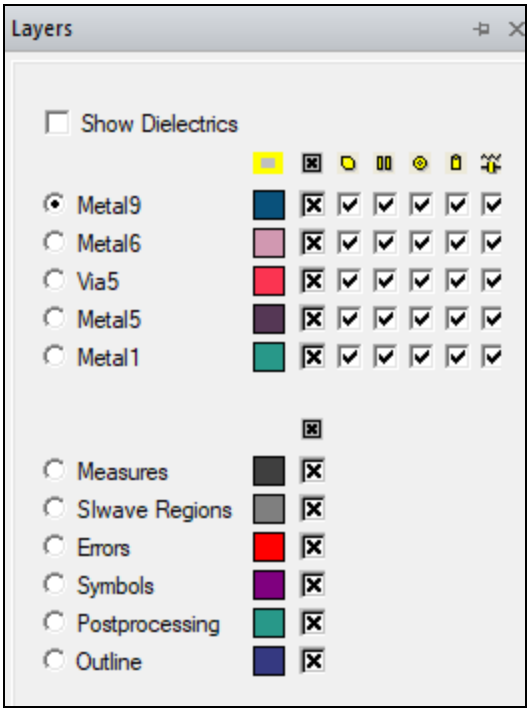
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Removing Small Lines in the Layout Editor

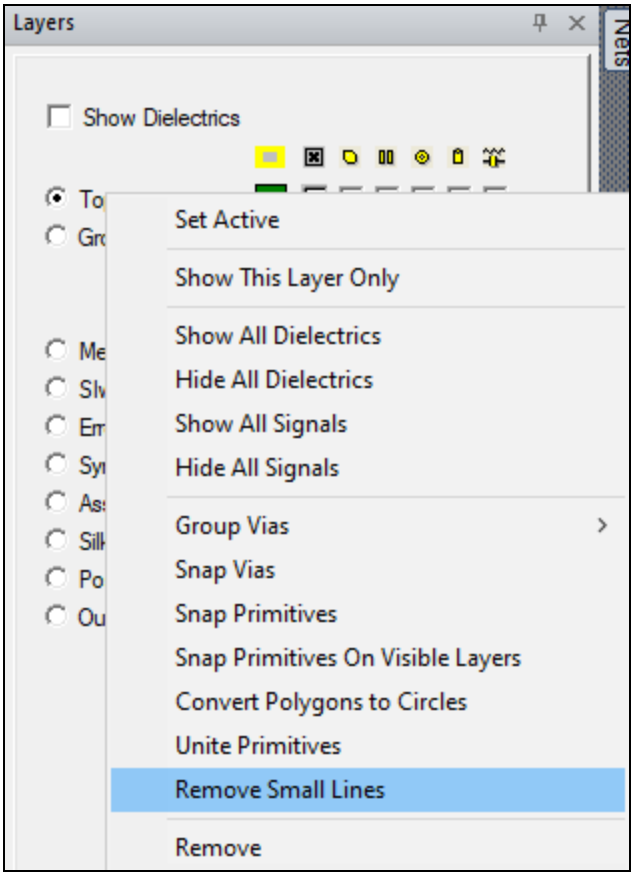
1. If the **Layers** window is not visible, navigate to the **Layers** tab. Click the **Layers** tab to open the window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



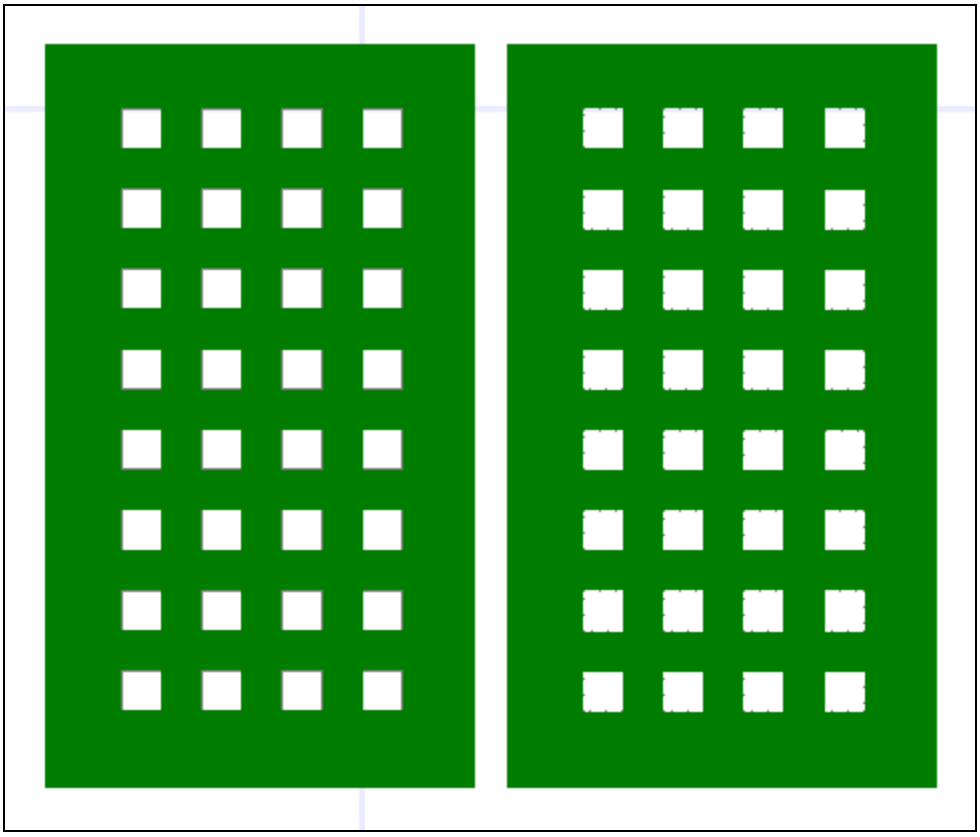
2. Right-click the selected layer (e.g., **Top**) and select **Remove Small Lines**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Lines in the designated layer are compared to the value in the **Tolerance** field. Any line with the same total area or smaller than the set **Tolerance** is removed 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.

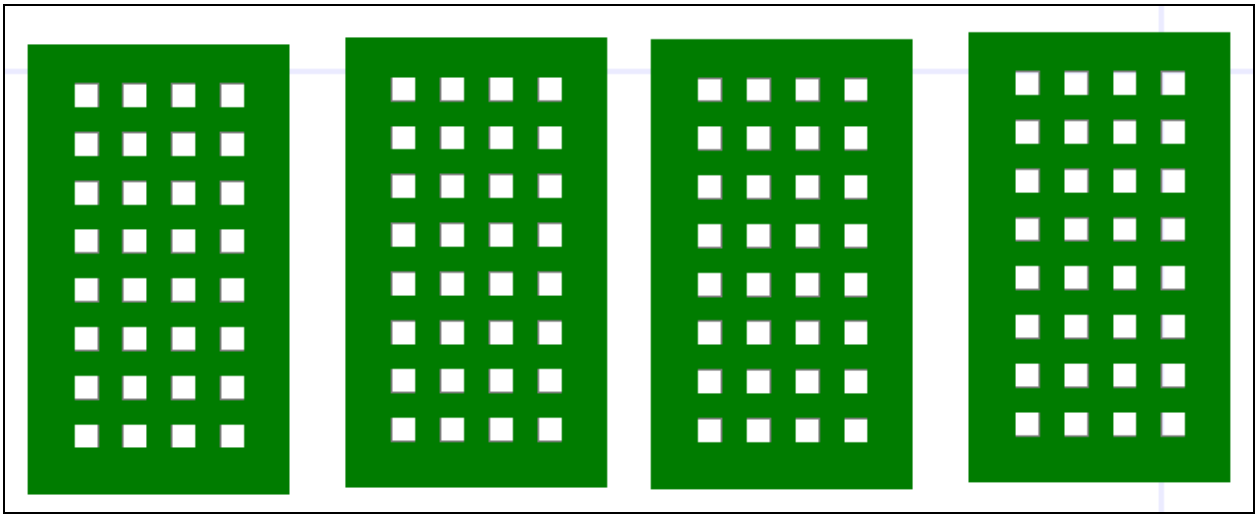


Removing Small Holes in IC Mode

Designs can contain inappropriate holes that users may want to remove. Users can clean up a design by using the **Remove Holes** feature. Holes are removed if their total area is smaller than the threshold set by the user, unless otherwise excluded using the parameters in the **Remove Small Holes** window.

Complete the instructions in the following subsections to adjust the **Remove Small Holes** settings, then remove small holes from a design in several ways.

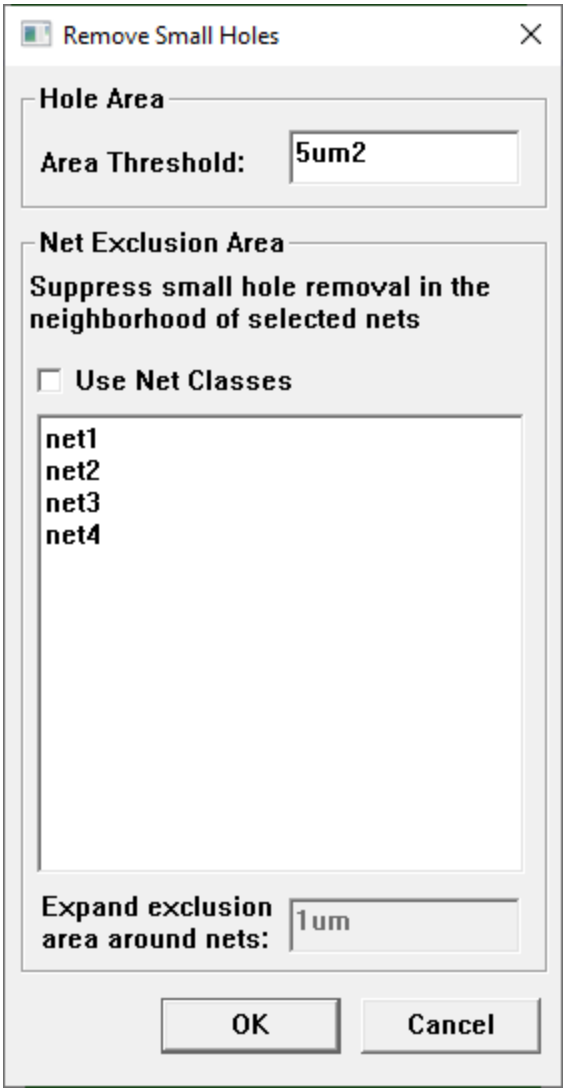
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Accessing the Remove Small Holes Window

From the **Layout** ribbon, click **Remove Holes** to open the **Remove Small Holes** window.

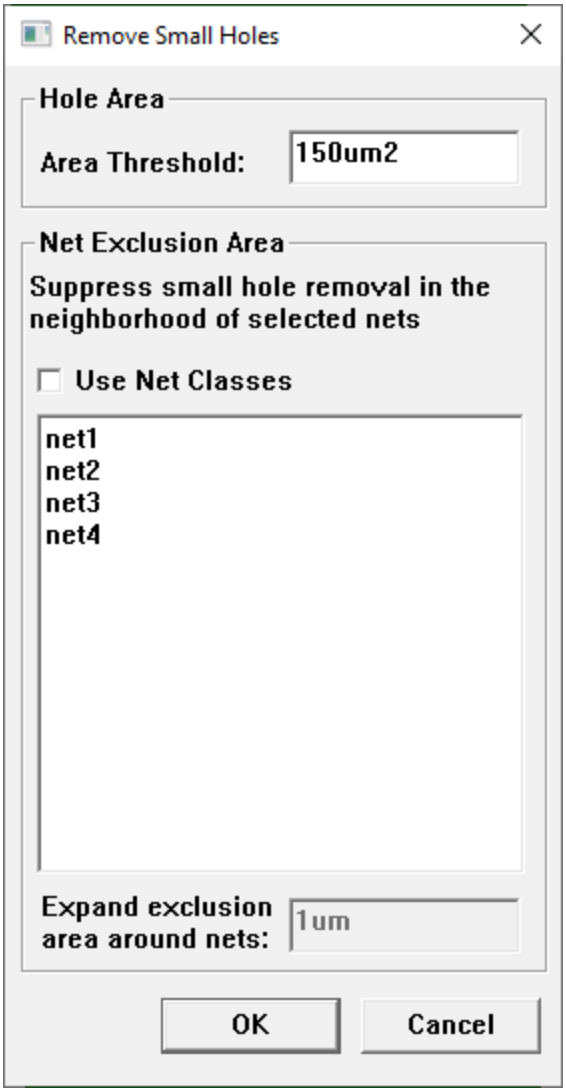
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Removing Small Holes Using Area Threshold

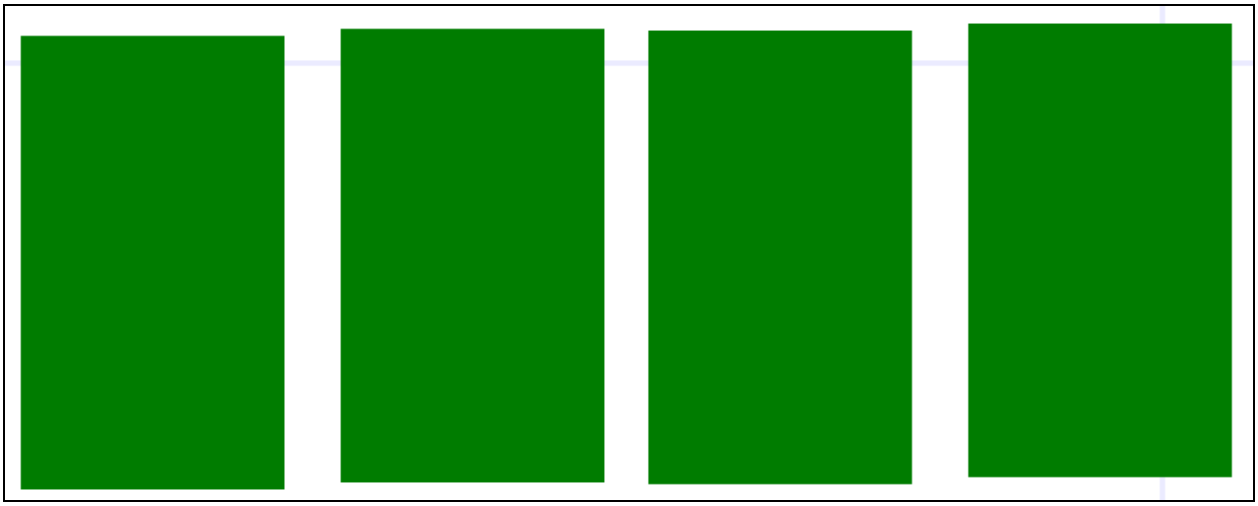
1. From the **Hole Area** area, enter a value in the **Area Threshold** field (e.g., **150um2**, although the default value is **5 um^2**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



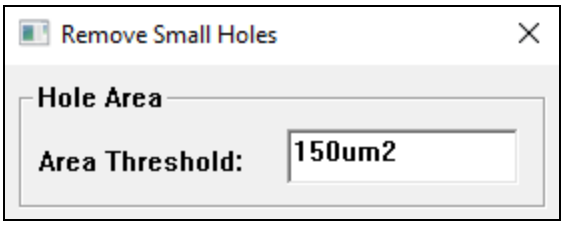
2. Click **OK** to close the **Remove Small Holes** window. All of the holes are compared to the value in the **Area Threshold** field. Any hole the same size or smaller than the entered value is removed from the design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Removing Small Holes Using Net Exclusion

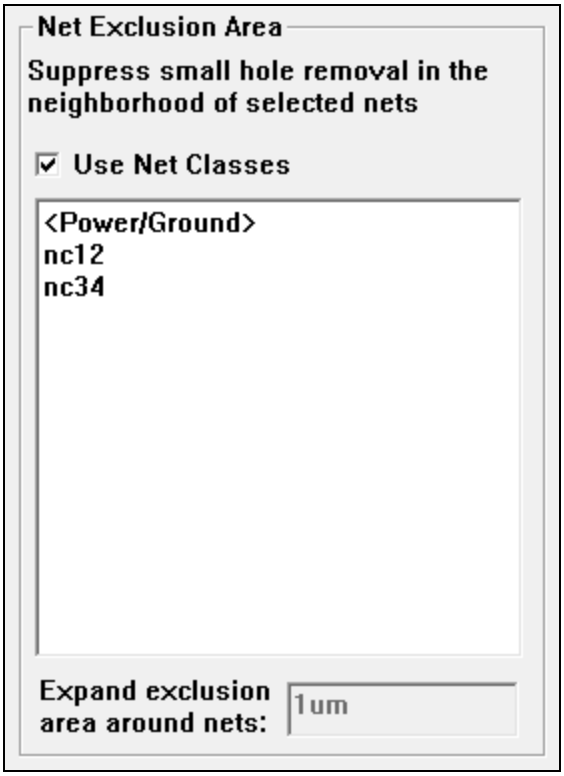
a. From the **Hole Area** area, enter a value in the **Area Threshold** field (e.g., **150um2**).



b. From the **Net Exclusion Area** area, do the following:

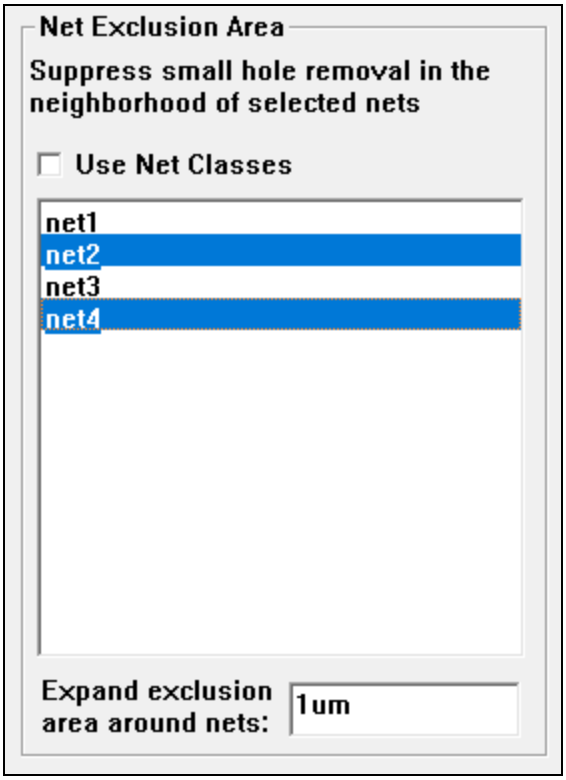
- If required, check the **Use Net Classes** box to select from a list of net classes rather than individual nets.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

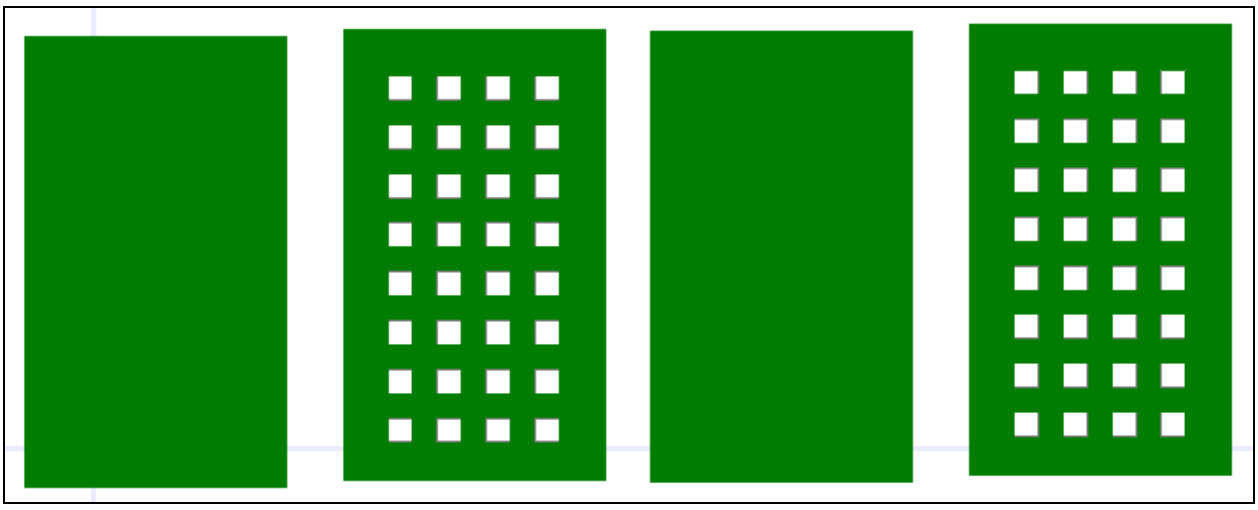


- Click a net/net class to exclude from hole removal. **Ctrl+click** to select more than one net/net class (e.g., **net2** and **net4**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



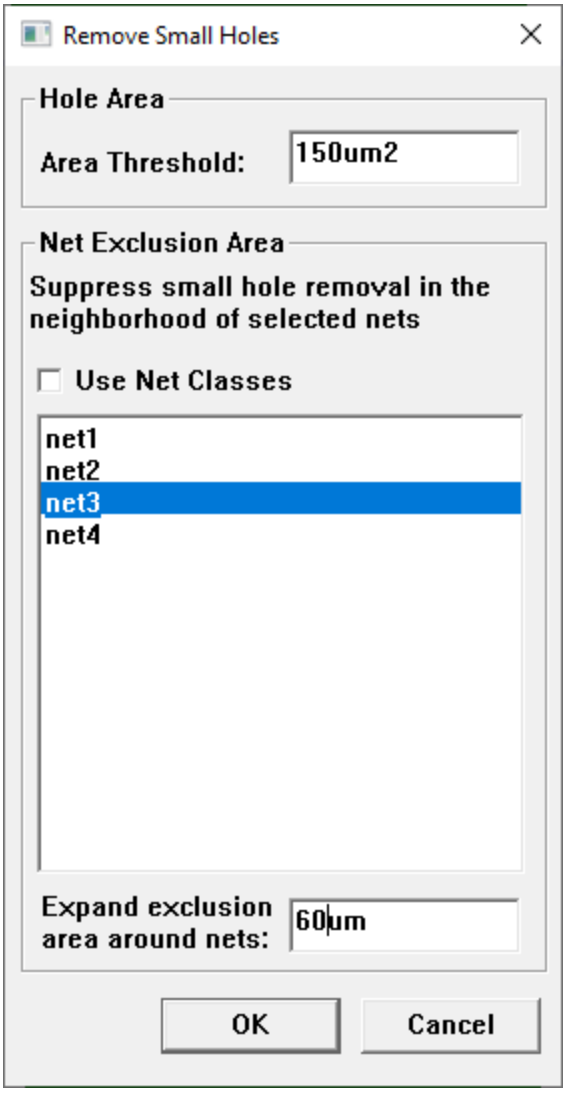
c. Click **OK** to close the **Remove Small Holes** window. All of the holes that are **not** in exclusion zones are compared to the value in the **Area Threshold** field. Any hole the same size or smaller than the entered value is removed from the design.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

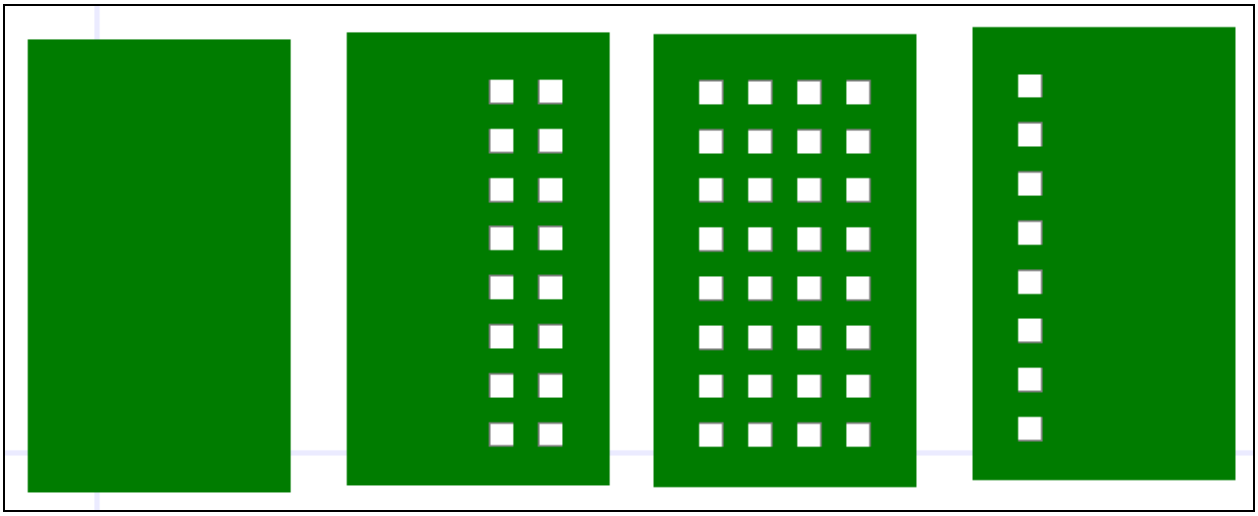
Expanding the Exclusion Area

After selecting nets or net classes to exclude from hole removal (e.g., **net3**), the **Expand exclusion area around nets** field is enabled. Enter a value in the **Expand exclusion area around nets** field (e.g., **60um**) to widen the exclusion area around the selected net(s)/net class (es).



Click **OK** to close the **Remove Small Holes** window. All of the holes that are **not** in exclusion zones, as defined by user-selection in the **Net Exclusion Area** area list and the **Expand exclusion area around nets** field are compared to the value in the **Area Threshold** field. Any hole the same size or smaller than the entered value is removed 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.



Cropping and Removing Small Fragments in IC Mode

Crop a design to filter nets and/or clip a design by a polygon. If cropping a design leaves behind inappropriate fragments, remove them by adjusting the selected parameter in the **Geometry Simplification Settings** window. Before cropping, consider the following:

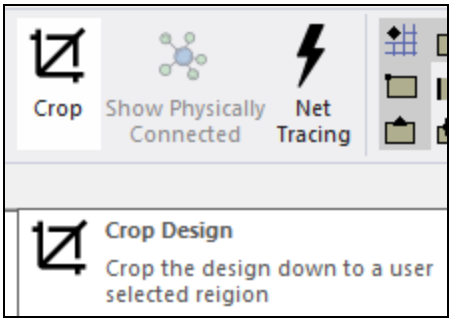
- If a polygon is selected before the operation is performed, it becomes the default extent polygon.
- If the selection is made entirely by net, then no geometry should be selected before invoking the operation. If multiple polygons are selected, the union of these polygons forms the extent.
- If no data is selected, a new design is created with a stackup identical to the source design, but no geometry is transferred.
- **Crop** works on hierarchy and footprints, however, everything is **flattened** into the sub-design that is created.

Cropping a Design

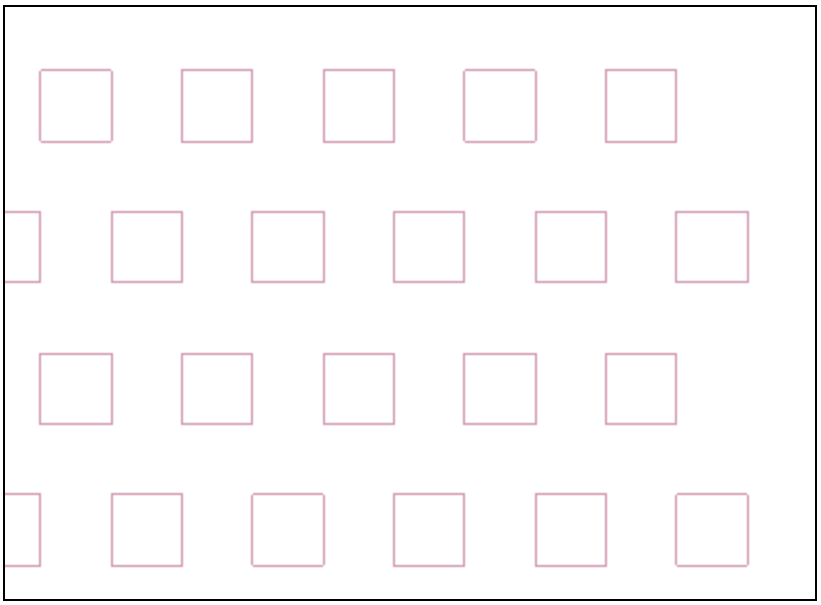
Follow these steps to crop a design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Layout** ribbon, click **Crop**.

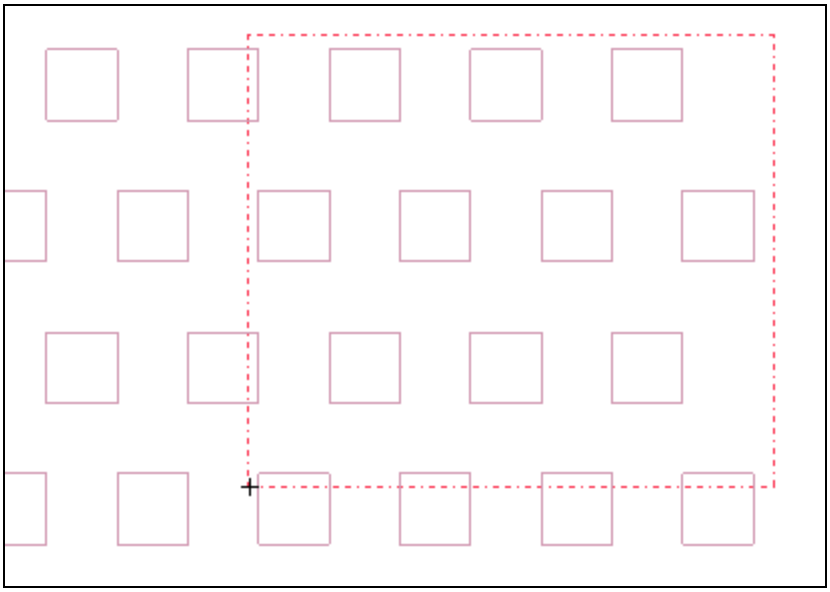


2. If applicable, navigate to the **Layout Editor** tab and the selected components.

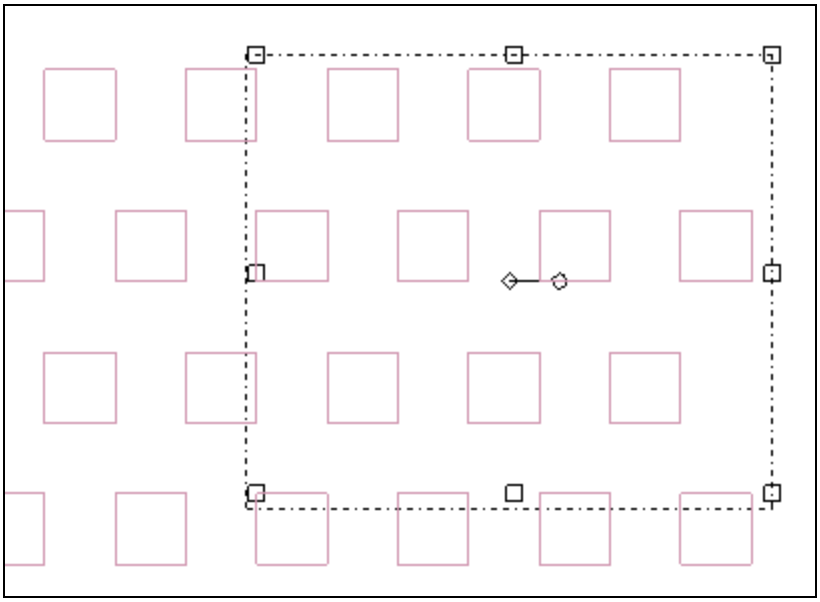


3. **Click** within **Layout Editor** to begin selecting a region of the design to preserve. Move the cursor to surround the entire region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

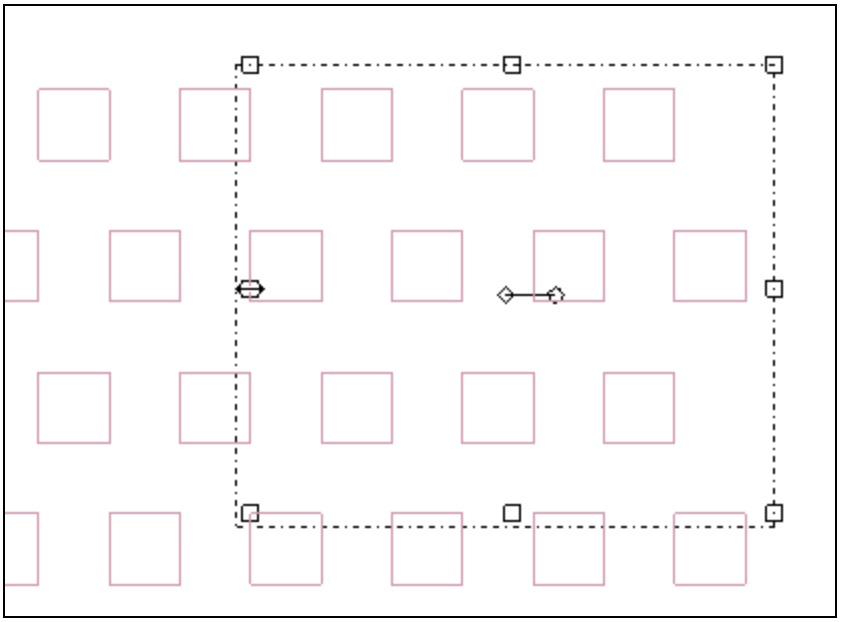


4. **Click** to define the region within the bounding box.

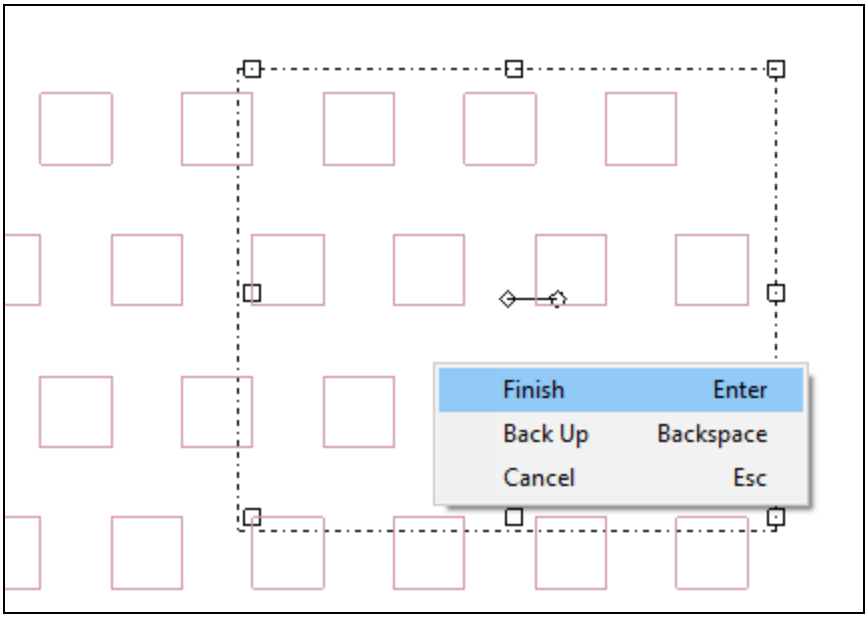


5. If applicable, expand or contract the bounding box. Hover over any of the square handles and the cursor will change into a double-headed arrow. Then **click+hold** the handle and drag to change the size of the selected region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

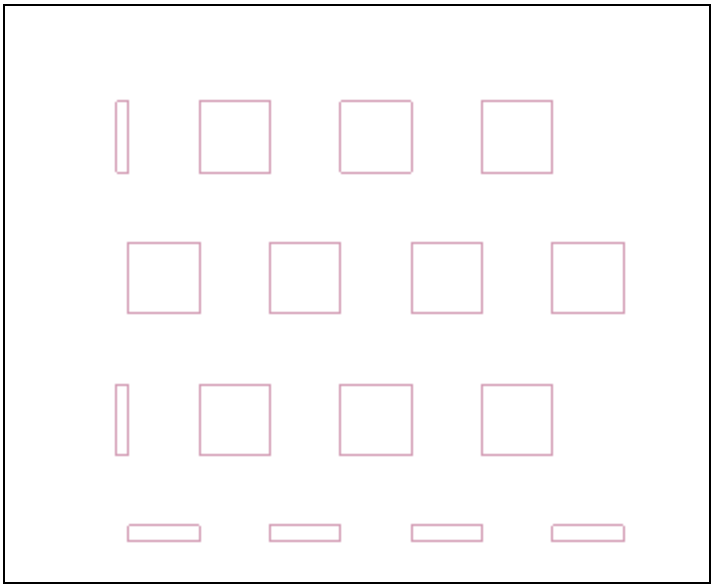


6. Right-click anywhere in the **Layout Editor** and select **Finish**.



Every element outside of the selected region is removed from the design. From the following screenshot there are fragments left behind by the **Crop**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



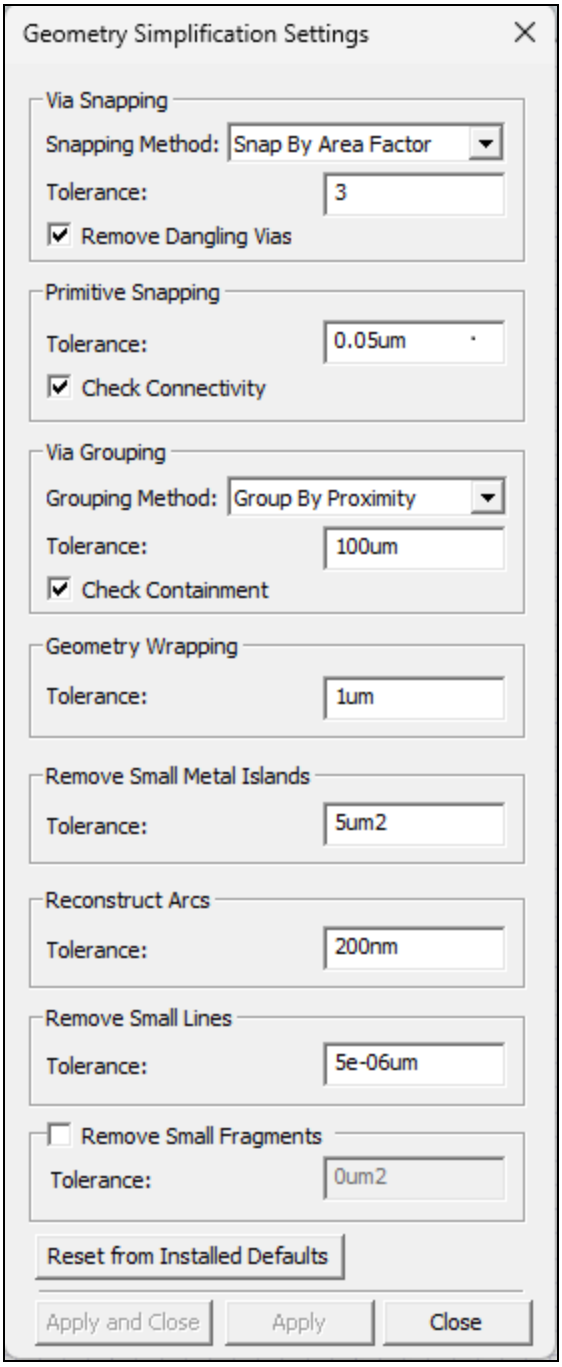
Removing Small Fragments From a Design

Follow these steps to configure **Remove Small Metal Fragments**, then remove fragments within a design.

1. From the **Layout** ribbon, click **Settings**.

The **Geometry Simplification Settings** window opens.

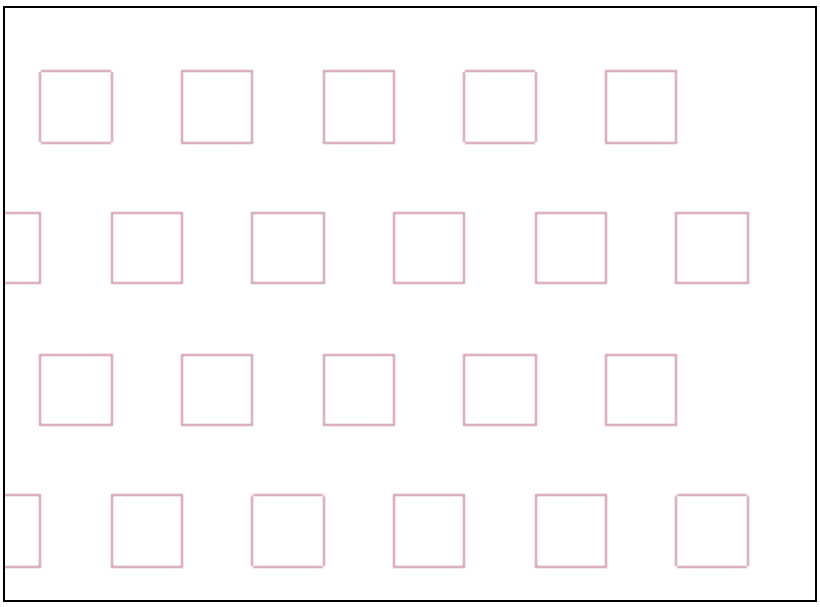
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



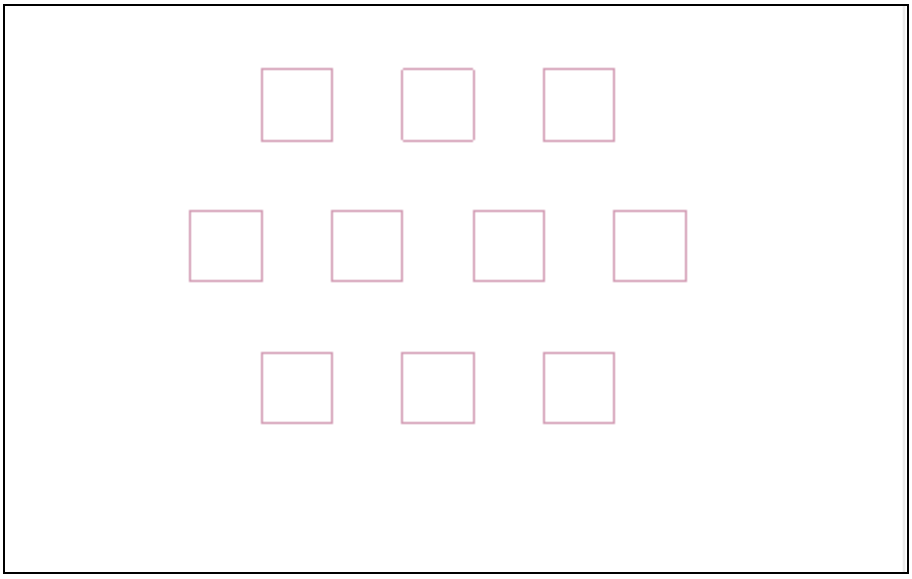
2. From the **Remove Small Fragments** area, do the following:
    a. Check the **Remove Small Fragments** box to activate the feature.
    b. Enter a value in the **Tolerance** field (e.g., **25um**, although the default is **5um**).
3. Click **Apply and Close**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Ctrl**+**Z** to reset the last changes made in the **Layout Editor** (e.g., undo the crop performed in Cropping a Design).



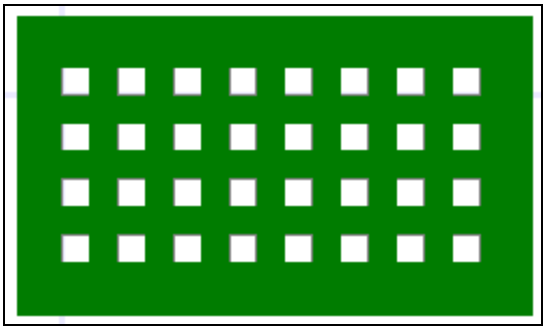
5. Repeat steps **3-6** in Cropping a Design. The fragments left behind by the **Crop**were at or below the **Tolerance** value, so they are also removed.



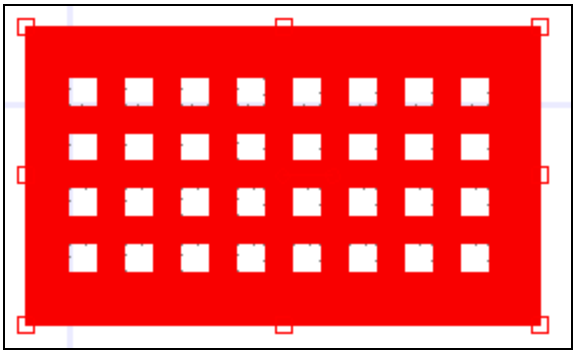
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying and Pasting in IC Mode

Design elements can be copied and pasted in IC mode, just like in General mode. Follow these steps to copy and paste in IC mode.

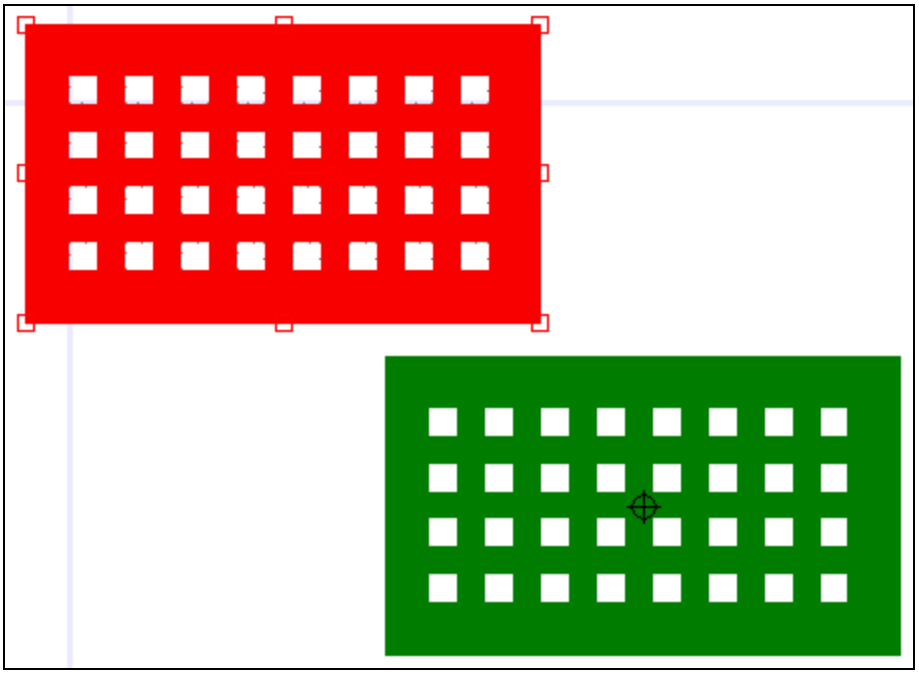


1. From the **Layout Editor**, select the appropriate element(s) and press **Ctrl**+**C**.

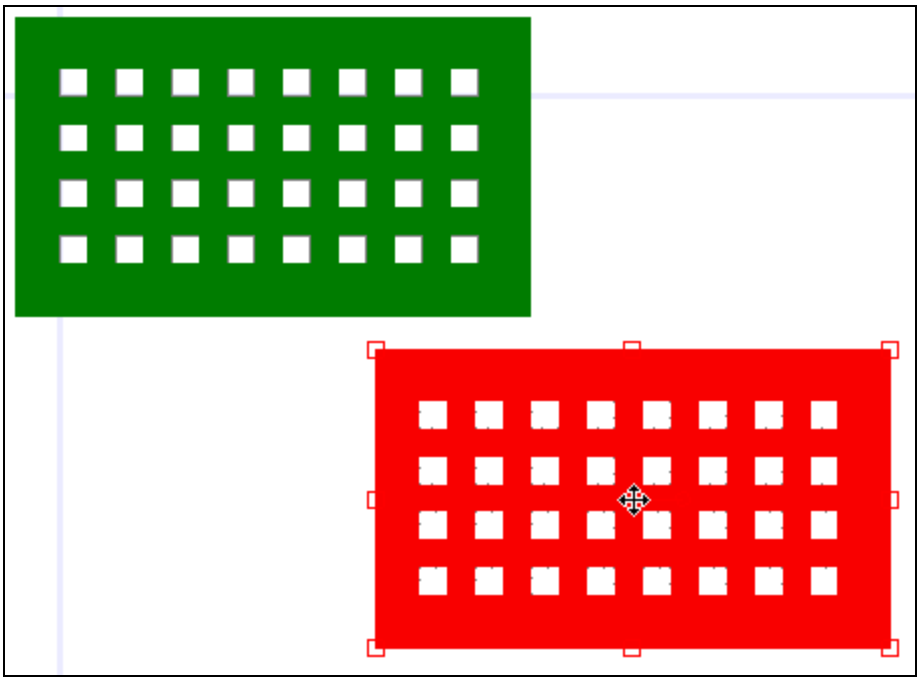


2. Press **Ctrl**+**V** to create a copy of the element. The copy is temporarily locked to the cursor.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

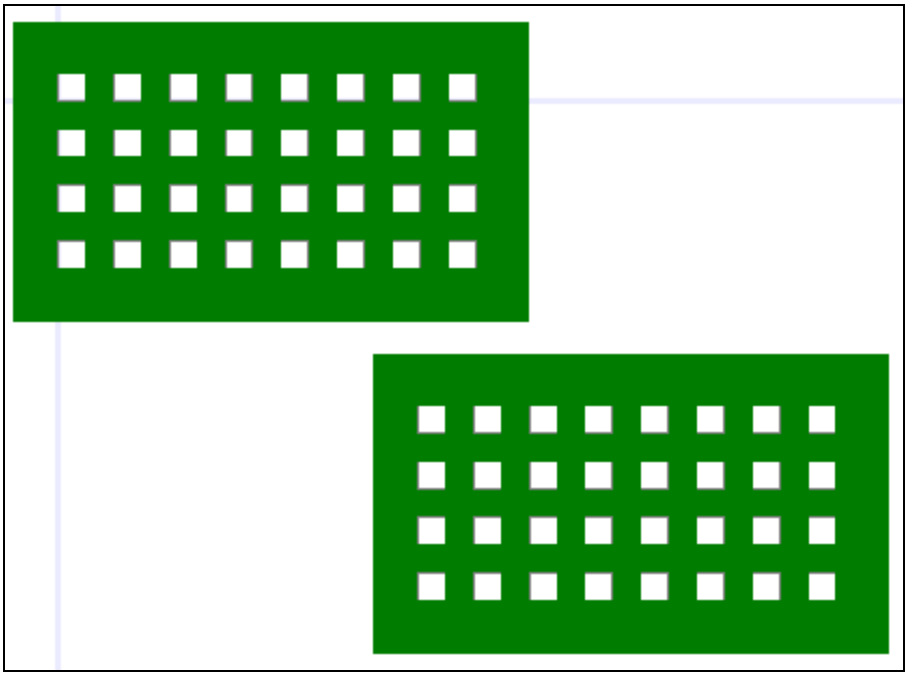


3. Choose an area in the **Layout Editor** to place the copy. **Click** to set the copy down.



4. Click anywhere else in the **Layout Editor** to deselect the copy.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For other instructions on the usage of the **Layout Editor**, keyboard commands, and simple tasks like copying and pasting, refer to Getting Started with Ansys Electronics Desktop.

Working with Objects

The following topics describe various tools and methods for working with objects in the Layout Editor.

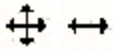
Selection Modes in the Layout Editor

Three selection modes exist: **General**, **Handle**, and **Edge**.

- General Selection Mode — Used primarily to select objects. **General Mode** is the default section mode. You can activate the General selection mode by selecting **Edit > Select** or by clicking the Select Elements icon .
- Handle Selection Mode — Used to select object handles. You can activate the Handle selection mode by selecting **Edit > SelectHandles** or by clicking the Select Handles icon .
- Edge Selection Mode — Used to select object edges. You can activate the Edge selection mode by selecting **Edit > SelectEdges** or by clicking the Select Edges icon .

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 selection mode, the cursor becomes a small cross hair when it is not positioned over a selectable object:

When the cursor is positioned over a selectable object, or the handle of a selectable object, the cursor changes to one of the following to indicate what object/handle can be moved: 

To give a visual clue as to the current selection mode, the cursor changes in the following manner:

	Not over selectable Edge/Handle	Over selectable Edge/Handle
General Selection Mode		
Edge Selection Mode		
Handle Selection Mode		

Using Handles

Handles are graphical attachments to objects that allow you to manipulate objects in the Layout editor. Two general types of handles exist: **Implicit** and **Explicit**. Implicit handles are created automatically, while Explicit handles are created by the user.

Implicit Handles

- In Layout, every primitive object (void, line, polygon) and compound object (via, footprint, sub-circuit) contains **Implicit** handles — identified by small red squares — which can be used to resize the object. There are two kinds of Implicit handles: **Edge** handles and **Point**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

handles. Edge handles pertain to lines and polygons, and can be used to drag an edge. Point handles pertain mainly to points, and can be dragged/deleted separately (but Point handles can also be inserted into the edges of lines and polygons). Edge and Point handles can be selected by clicking upon them, or by using the Handle Select Tool.

- Every Layout object contains an implicit **Center** handle — identified by two small red circles connected by a line — which can be used to rotate or drag the object.
- Selections containing multiple objects have one Implicit handle which is placed at the geometric center of the data.

Explicit Handles

- Footprints may contain **Explicit** handles — identified by small black squares — that are created by users and can be used to drag or resize the footprint.

Selecting Handles

The **Select Handles** command on the **Draw** menu allows you to select just the handles of selected items in the **Layout Editor**.

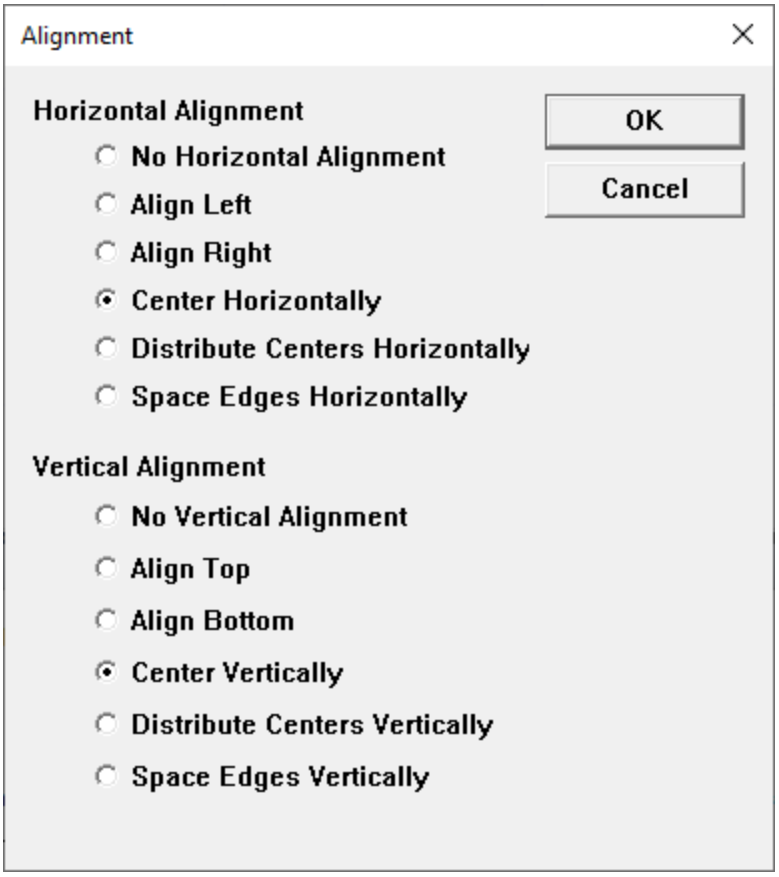
1. From the **Draw** menu, select **Select**. Then click the object you want to manipulate.
2. From the **Draw** menu, select **Select Handles**. The handles on the selected items appear as squares.
3. From the **Layout Editor**, move over the handle you want to select. A square appears at the cursor, indicating that you are over the handle.
4. Click the handle to select it.

The selected handle is highlighted.

Aligning Objects in the Layout Editor

To align objects in the **Layout Editor**, first select more than one object. From the **Draw** menu, select **Align** to open the **Alignment** window.

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 *Horizontal Alignment* controls are available:

- **No Horizontal Alignment**
- **Align Left**
- **Align Right**
- **Center Horizontally**
- **Distribute Centers Horizontally** places polygons so the horizontal spaces between their centers are equal
- **Space Edges Horizontally** places multiple polygons so the horizontal spaces between their edges are equal

The following *Vertical Alignment* controls are available:

- **No Vertical Alignment**
- **Align Top**
- **Align Bottom**
- **Center Vertically**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Distribute Centers Vertically** places multiple polygons so the vertical spaces between their centers are equal
- **Space Edges Vertically** places multiple polygons so the vertical spaces between their edges are equal

Auto-Aligning Objects

Auto-alignment of components is available by selecting one or more objects and clicking **Ctrl**+**M**. But auto-alignment only rotates/positions components to match connections, and does not flip or unflip them. Therefore, auto-aligning components which are flipped or have twisted connections will align the components improperly.

Selecting Overlapping Objects in the Layout Editor

When two or more objects in the **Layout Editor** overlap (i.e., are selectable on the same location) click the location and type "**b**" to cycle through the selectable objects. (Alternatively, click **Cycle Selection** on the **Edit** drop-down menu.) This highlights each object as it is selectable and allows you to select from objects that overlap.

Selecting Edges in the Layout Editor

The **Select Edges** command on the **Edit** menu enables you to select just the edges of items in the **Layout Editor** and measure the length of the edges selected.

1. Do one of the following:
    - From the menu bar, click **Edit** > **Select Edges**.
    - From the Layout ribbon, open the **Select Elements** drop-down menu and click **Select Edges**.
    - Right-click in the **Layout Editor** and click **Select Edges**.
2. From the **Layout Editor**, move over the edge you want to select. A line appears at the cursor, indicating that you are over that edge.
3. Click the edge to select it.

The selected edge is highlighted. The **Distance** field on the Status bar displays the length of the edge. If you select multiple edges, the **Distance** field shows the sum of the edges' lengths.

Grouping Objects in the Layout Editor

You can group objects in the **Layout Editor** by using the Coordinate System (CS) commands listed beneath the Draw menu. To group selected objects together, select **Draw** > **Group into CS**. To ungroup objects, select **Draw** > **Ungroup CS or 3D Structure**. For more information see Coordinate Systems.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

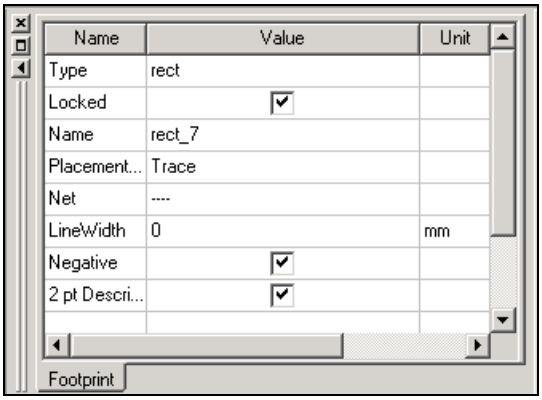
Negative Objects in the Layout Editor

Objects may be designated as a negative. The negative attribute is accessed through the **Properties** window or an object's **Propertywindow**. The following are defined as negative objects:

- Remove material from all items they overlap.
- Export to Gerber, GDSII and DXF.
- Can be copied to the clipboard.

A negative object on a ground layer does not remove material on the layer itself, but rather, interacts with other primitives in the same way as on other layers. Negative objects may contain voids. But a negative always removes material and never adds it. Consequently, a void in a negative is merely an area where material is not removed.

To designate an object as a negative in the **Layout Editor**, in the **Properties** window or the object's **Properties** window (double-click the object) check the **Negative** selection box.



Position Locking

Primitives, components, vias, and pins can all be locked into place in the **Layout Editor**. The locking attribute is accessed through the **Properties** window or an object's **Properties** window.

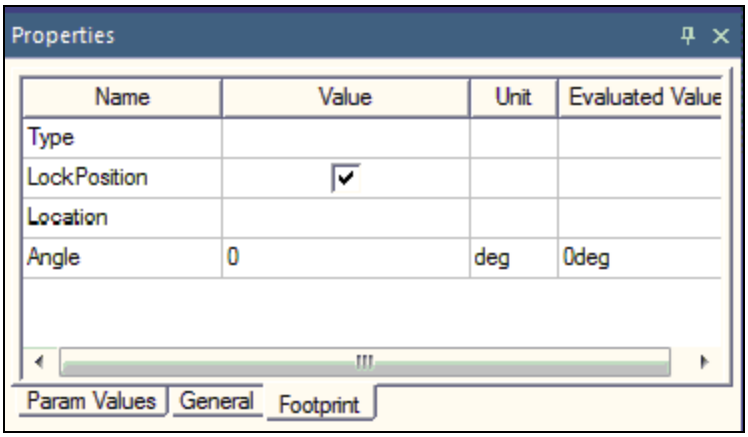
Locked objects do not move during an **Align** or **Microwave-port Align** operation. If multiple objects are locked, the first pinned object encountered in the selection (based on the selection order) is used as an "anchor" during an align (if there are none, the first object is used). If there is only one object locked, that object becomes the origin of the alignment.

- Once locked, an object cannot be dragged or rotated.
- When locked, no object properties that change position or shape are editable.
- When a locked object is selected, the object handles are not shown.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Locking a primitive locks the voids it owns.
- Locking a void prevents it from moving when the owner is moved.

To lock an object in the **Layout Editor**, in the **Properties Window** or the object's **Propertieswindow** (double-click the object) check the **LockPosition** selection box.



Modifying Objects in the Layout Editor

You can quickly modify the position, dimensions, and other characteristics of objects created in the **Layout** window.

Deleting Objects in the Layout Editor

Follow these steps to delete objects in the **Layout Editor**.

1. Select objects to delete.
2. From **Edit**, select **Delete**, or press **Delete**.

Flipping Objects in the Layout Editor

- **Draw** > **Flip about Y axis** to flip the object horizontally.
- **Draw** > **Flip about X axis** to flip the object vertically.

Rotating Objects in the Layout Editor

You can rotate one or more objects by dragging the red rotation handle at the center 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.

1. Select the object you want to rotate.
2. Drag the circular part of the rotation handle in the direction you want to rotate the object.

> **Note:**
>
> To drag more than one object, select the objects and the rotation handle appears at the center point between the objects.

> **Note:**
>
> To rotate without re-wiring (“rubberbanding”), hold down the <Shift> key as you rotate the object(s).

Uniting Objects in the Layout Editor

To join two or more objects into one object, use the **Layout > Merge Polygons > Union** 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 **Layout** menu, select **Merge Polygons** > **Union**.

The objects are united.

Union and **Intersection** operations support using selected objects on multiple layers, and each layer object is operated on separately. However, **Subtraction** is only allowed if the selected objects are on the same layer. Selected objects may be negative objects.

- If only negative objects are selected, the merge operation is performed the same as with positive objects, but the resulting objects remain negative objects.
- If both negative objects and positive objects are selected:

  — Complete the union/intersection/subtraction operation on the positive objects

  — Always union the negative objects

  — Subtract the negative object result from the positive object result

> **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 the following:
>
> - Copy the objects, then paste them back into the design after uniting them.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subtracting Objects in the Layout Editor

1. Select the object from which you want to subtract other objects.
2. Select the objects you want to subtract.
3. From the **Layout** menu, select **Merge Polygons** > **Subtract**.

The new object retains the name, color, and material of the first object selected.

Union and **Intersection** operations support using selected objects on multiple layers, and each layer object is operated on separately. However, **Subtraction** is only allowed if the selected objects are on the same layer. Selected objects may be negative objects.

- If only negative objects are selected, the merge operation is performed the same as with positive objects, but the resulting objects remain negative objects.
- If both negative objects and positive objects are selected:
    - Complete the union/intersection/subtraction operation on the positive objects
    - Always union the negative objects
    - Subtract the negative object result from the positive object result

> **Note:**
>
> By default, the objects being subtracted on the first object selected are *not* preserved for later use. If you want to keep a copy of the objects being subtracted on the first object selected, do the following:
>
> - Copy the objects, then paste them back into the design after subtracting them.

Creating Objects from Intersections in the Layout Editor

To create a new object on the intersection of two or more objects, click the **Layout > Merge Polygons > Intersection** command.

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.

2. From the **Layout** menu, select **Merge Polygons** > **Intersection**.

The original objects vanish, leaving only the new object that was formed on 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.

Union and **Intersection** operations support using selected objects on multiple layers, and each layer object is operated on separately. However, **Subtraction** is only allowed if the selected objects are on the same layer. Selected objects may be negative objects.

- If only negative objects are selected, the merge operation is performed the same as with positive objects, but the resulting objects remain negative objects.
- If both negative objects and positive objects are selected:

  — Complete the union/intersection/subtraction operation on the positive objects

  — Always union the negative objects

  — Subtract the negative object result from the positive object result

> **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 the following:
>
> - Copy the objects, then paste them back into the design after creating the new object on the intersection.

Duplicating Objects in the Layout Editor

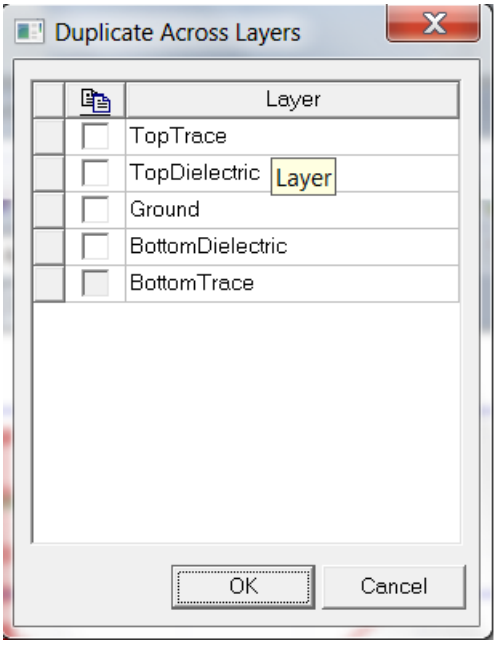
Duplicate Across Layers

The **Duplicate Across Layers** command supports duplicating layout objects to other layers at the same location. It works for both layout editing and footprint editing. The same command is available in the shortcut menu. The following layout objects are eligible for duplication:

- **Primitives**: Arc, Circle, Line, Rectangle, Polygon. A void cannot be duplicated alone; instead, a void is duplicated along with the selected object to which it belongs.
- **Edge port**: If an edge port is selected and all its edges are attached to primitives on the same layer and all those primitives are also selected, the edge port are duplicated.

When you have layout objects selected and click **Draw** > **Duplicate** > **Across Layers,** a window opens for target layer selection. (If no suitable items are selected for duplication, a warning message is displayed instead.)

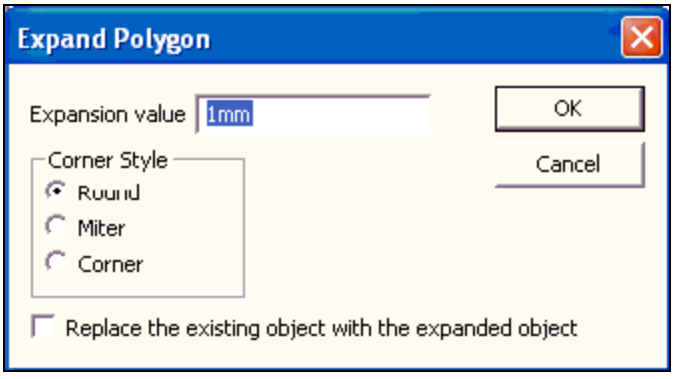
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Only stackup layers are listed as valid target layers. Locked layers are not eligible as target layers and are unavailable. If all items are on the same layer, that layer is also unavailable.
- Right-click the layer selection grid to select from all signal, ground, or dielectric layers.
- After you select target layers and click **OK**, the eligible items are duplicated to the layer(s) you select. All duplicated items remain selected if they are on layers that are selectable and visible. Selected items that are not eligible for duplication is listed in the **Message Manager** window.

Expanding Objects in the Layout Editor

You can “expand” objects in Layout using the **Draw > Expand** command. Click **Draw > Expand** to open the **Expand Polygon** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Expansion value** can be positive or negative (e.g., “2mm”, “-2mm”). The object is expanded or shrunk by the distance specified.
- The **Corner Style** (where expanded sides meet) can be specified as **Round**, **Miter**, or **Corner**. When **Corner** is selected, if the corner extension results in a projection greater than 8 times the expansion value (which corresponds to angles approximately 15 degrees or less), the two meeting sides are truncated to reduce the corner projection.
- You may either **Replace the existing object with the expanded object** or, by default, create a new object.

Paste Objects In Place in the Layout Editor

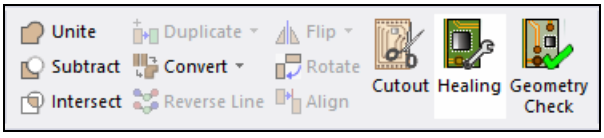
Sometimes it is convenient to copy an object and paste it without an X,Y offset or displacement. For instance, when creating copies of parameterized objects, or when copying an object to the same location but in a different layer.

1. Select the object(s) to be duplicated.
2. Choose **Copy** on the right-click menu or on the **Edit** drop-down menu.
3. Select the layer you want to draw upon on the **ActiveLayer** drop-down menu on the **Layout** toolbar.
4. Choose **Paste in Place** on the shortcut menu or on the **Edit** drop-down menu and the object is pasted onto the active layer using the same X,Y coordinates from which it was copied.

Geometry Healing in the Layout Editor

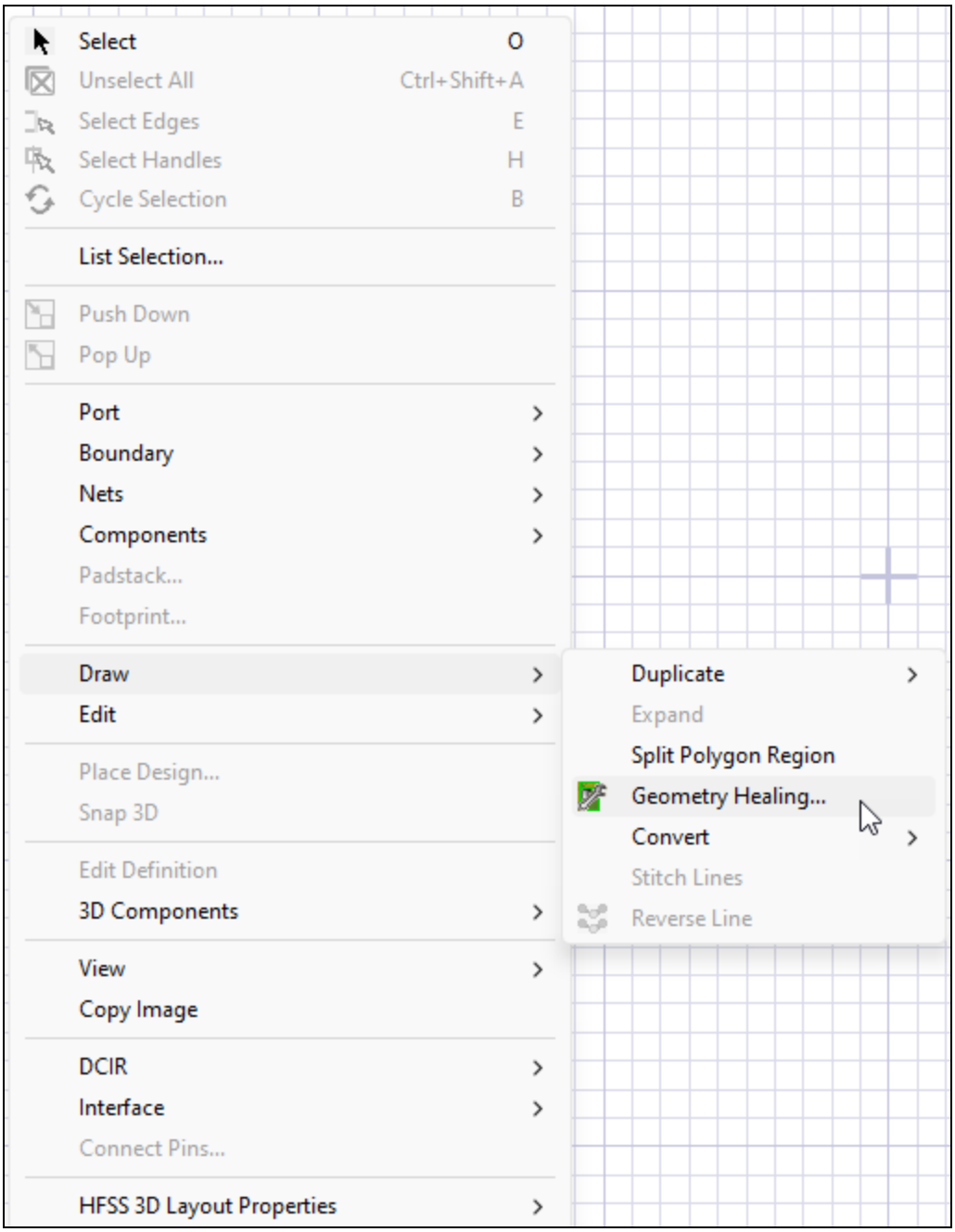
To access the **Geometry Healing** window, do either of the following:

- Navigate to the **Layout** ribbon and click **Healing**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

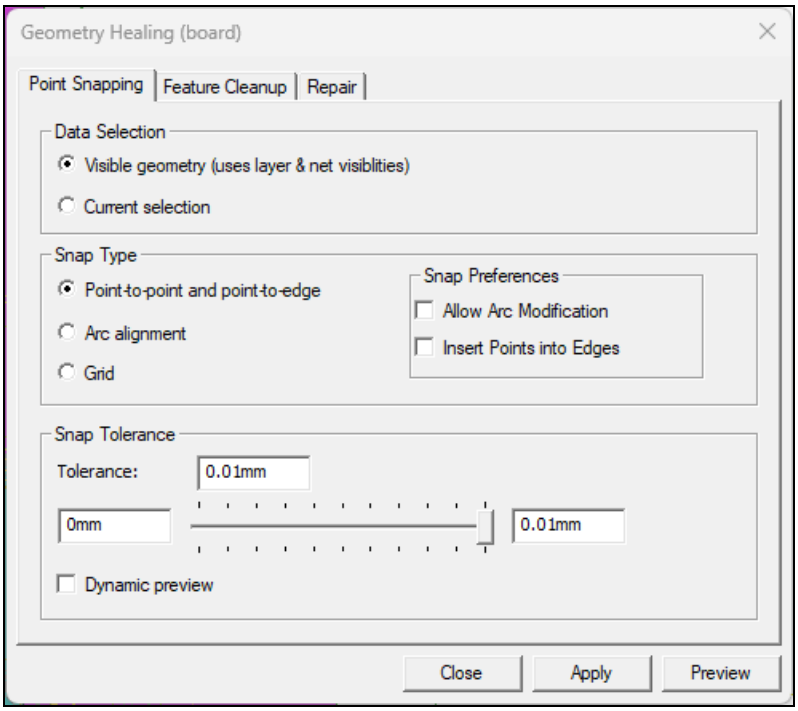
- Right-click in the **Layout Editor** and select **Draw** > **Geometry Healing**.



The **Geometry Healing** window opens at the **Point Snapping** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Point Snapping



From the **Point Snapping** tab, do the following:

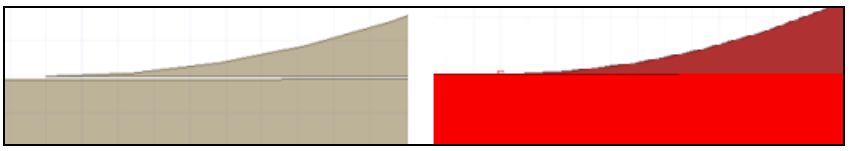
- **Data Selection** – select either of the following options to determine what geometry to heal:
  - **Visible geometry (uses layer & net visibilities** – analyzes all visible geometry, based on current layer and net visibility settings.
  - **Current selection** – analyzes only the currently selected geometries.
- **Snap Type** – select one of the following snap types:
  - **Point-to-Point and point-to-edge** – attempts to snap points to points and points to edges. Keep in mind:
    - only points in the tolerance are moved.
    - points are snapped across nets (i.e., nets are not shorted).
    - a snap is rejected if the angle around the point changes by more than 1 degree.
    - locked objects can be snap targets but are not themselves altered.
    - points may snap to the edge of a trace but traces themselves are not moved.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

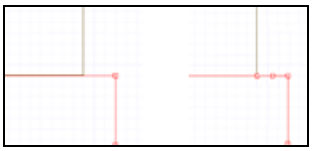
- vias act as snap targets but are not changed.
- rectangles may become polygons.

- **Arc Alignment** – aligns arcs that share a similar center or a similar radius. Keep in mind:
  - only arcs with centers or radii in the tolerance are adjusted.
  - healing aligns the start/end points of traces and bondwires with circles and vias.
  - vias are treated as fixed objects (i.e., they do not move, but other arcs may align to them).
  - healing attempts to align arcs but does not guarantee alignment, especially when multiple arcs are connected in a polygon.
- **Grid** – snaps all geometry points to a grid defined by the user-specified tolerance. All points (i.e., coordinates) are rounded to the tolerance value supplied. Keep in mind:
  - healing snaps all primitives.
  - healing snaps the center location of vias.
  - healing snaps the start/end locations of bondwires.

- **Snap Preferences** – check either/both of the following options, as appropriate:
  - **Allow Arc Modification** – arcs drawn in close proximity to another primitive's edge will snap to the primitive if they are both healed.



  - **Insert Points into Edges** – if adjacent primitives with edges in close proximity are both healed, new points will appear at midpoints between their existing edges and at the new edges created where they snap together.

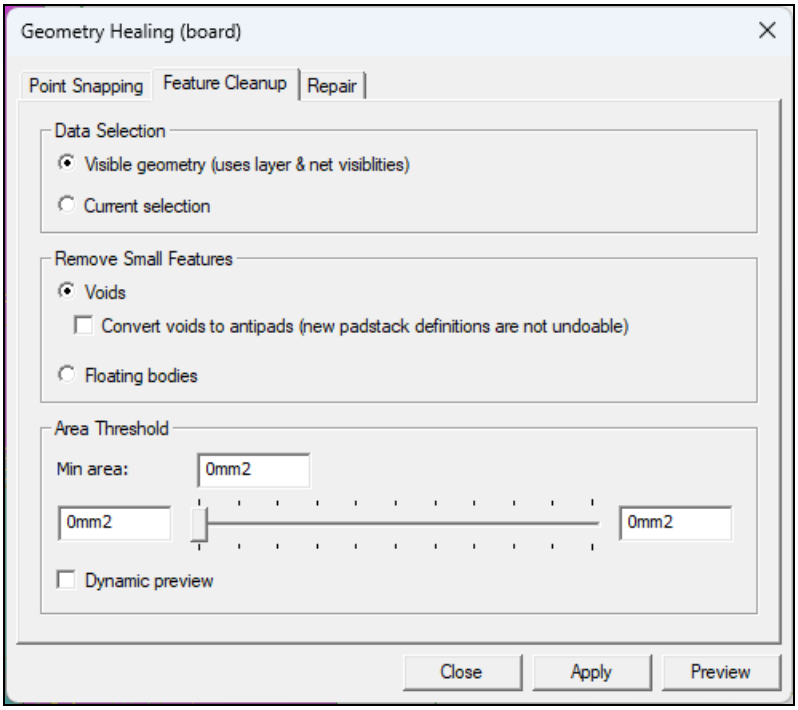


- **Snap Tolerance** – Enter a value in the first field to set a minimum area, or move the slider to an appropriate value. All points at a distance less than the specified distance are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

snapped. Check **Dynamic preview** to see a live update as the threshold is adjusted.

Feature Cleanup



From the **Feature Cleanup** tab, do the following:

- **Data Selection** – select either of the following options to determine what geometry to heal:
  - **Visible geometry (uses layer & net visibilities** – analyzes all visible geometry, based on current layer and net visibility settings.
  - **Current selection** – analyzes only the currently selected geometries.

    > **Note:**
    >
    > For **Voids** and **Floating bodies** cleanup, all the geometry is inspected to determine if the criteria for cleanup is met.

- **Remove Small Features** – select one of the following small feature types to remove:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

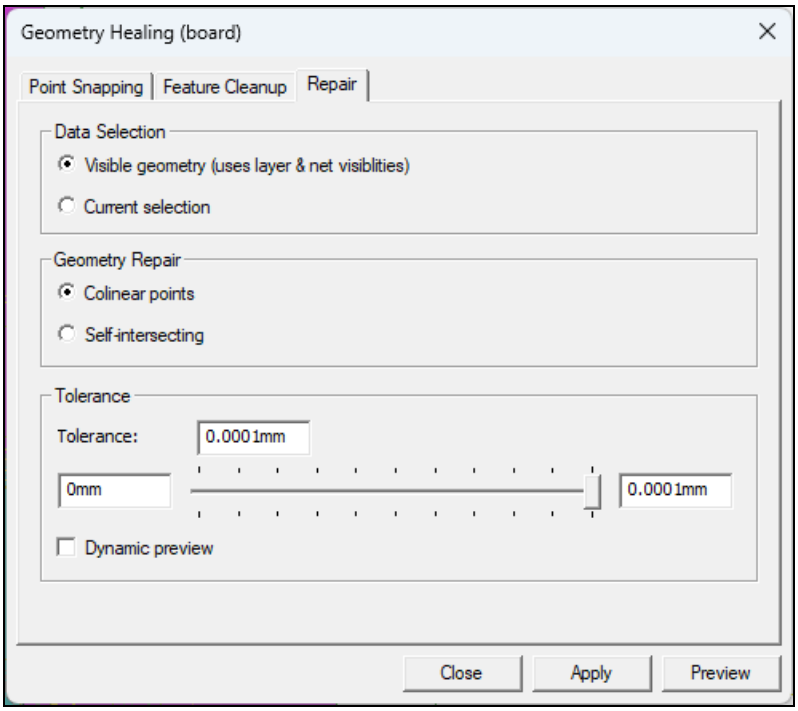
- **Voids** – removes voids with an area less than the user-specified **Area Threshold**. Keep in mind:
    - healing does not remove voids with vias passing through them.
        - If applicable, check **Convert voids to antipads (new padstack definitions are not undoable)**. Voids can only be converted if they meet the following criteria:
            - Voids can only be converted if they are circular or rectangular and the via passes through their centers.
            - The vias must be in a different net from the void owner (e.g., the via is in a TX net and the void owner is in GND).

            > **Note:**
            >
            > This creates new padstack definitions, which are not removed by **Undo**.

            If these conditions are met, the explicit void is replaced with an implicit void generated using an antipad.
- **Floating Bodies** – removes small islands of metal with an area less than the user-specified **Area Threshold**. Keep in mind:
    - if floating bodies are attached to other bodies, the total connected area is considered (takes into account vias).
    - during analysis, all geometry (selected and visible) is included.
    - healing does not remove bodies with ports attached.
- **Area Threshold** – enter a value in the first field to set a minimum area, or move the slider to an appropriate value. All features smaller than the specified area are removed. Check **Dynamic preview** to see a live update as the threshold is adjusted.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Repair



From the **Repair** tab, do the following:

- **Data Selection** – select either of the following options to determine what geometry to heal:
  - **Visible geometry (uses layer & net visibilities** – analyzes all visible geometry, based on current layer and net visibility settings.
  - **Current selection** – analyzes only the currently selected geometries.
- **Geometry Repair** – select either of the following geometry repair operations:
  - **Colinear points** – removes collinear points from polygons and trace center-lines. Keep in mind:
    - points are considered collinear if they deviate from the chord connecting the adjacent points by less than the user-specified **Tolerance**.
    - the analysis also removes very narrow intrusions and spikes from a polygon.
  - **Self-intersecting** – removes self-intersections from polygon data. Keep 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.

- the negative region of the self-intersection is excised.
- voids are correctly preserved – this may require duplicating a void if it overlaps two positive lobes of a self-intersecting shape.

- **Tolerance** – Enter a value in the first field to set a minimum area, or move the slider to an appropriate value. All points at a distance less than the specified distance are snapped. Check **Dynamic preview** to see a live update as the threshold is adjusted.

Once selections are made, select from the following:

Preview – shows a preview of the operation without changing the database. Various statistics appear after a preview or application. The worst case estimate is typically shown – click magnifying glass to jump to that location.

Apply – save changes.

Close.

Editing Arcs in the Layout Editor

You can edit an arc by changing its properties in the **Property Control** window or using the mouse as follows.

Move an arc in one of these ways:

- Drag the arc to the selected point.
- From the **Draw** menu, select **Select**. Select the arc, then type the center point's new coordinates in the **X** and **Y** fields.

Change an arc's end points in one of these ways:

- From the **Draw** menu, select **Select Handles**. Then select the start or end point handle, and drag it to the selected point.
- From the **Draw** menu, select **Select Handles**. Then select the start or endpoint and type the point's new coordinates in the **X** and **Y** fields.

Change an arc's height in one of these ways:

- Press **Ctrl** and drag the midpoint handle to the selected point.
- From the **Draw** menu, select **Select Handles**. Then select the midpoint handle and type the point's coordinates in the **X** and **Y** fields.

Adding Vertices to Edges in the Layout Editor

Add a new vertex to a line segment or polygon using the **Alt** key.

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 and the object handles become visible as squares on the object edges.
2. From the **Edit** menu, select **Select Handles** and the cursor becomes a square.
3. Hold down the **Alt** key. While holding down the **Alt** key, the cursor will take one of two shapes:

- While hovering over a polygon edge, the cursor becomes a diagonal line with a hollow circle to indicate the cursor **is** in a valid location to insert a vertex.
- While hovering anywhere else, the cursor becomes a diagonal line segment to indicate the cursor is **not** in a valid location to insert a vertex.

4. Click an appropriate point for the new vertex.
5. Repeat step 4 to insert additional vertices, as appropriate.
6. Release the **Alt** key and it becomes the square "Select Handles" cursor once again.
7. From the **Edit** menu, select **Select Handle** again to return the cursor to its original shape.

Working with Layers

The **Layout Editor** uses layers to organize and isolate sets of geometry and other visual indicators. Physical layers such as signal, via, dielectric, and negative signal layers compose the circuit board's stackup. Non-physical, conceptual layers display component layout, connectivity, etc. Physical layers must be created and added to the project, while most non-physical layers are created by default.

Layers are displayed and configured through the stackup editor. The stackup editor provides access to additional properties of the physical layers, such as material, thickness, and elevation. Geometrical information on these layers is used to generate masks for manufacturing. Stackups can be of three types:

- Laminate: Alternating signal and dielectric layers
- Multizone: Similar to laminate, but allows zones (areas) that contain a subset of all layers in the stackup
- Overlapping: Contains signal layers that can overlap vertically with other layers. Dielectric layers must be contiguous.

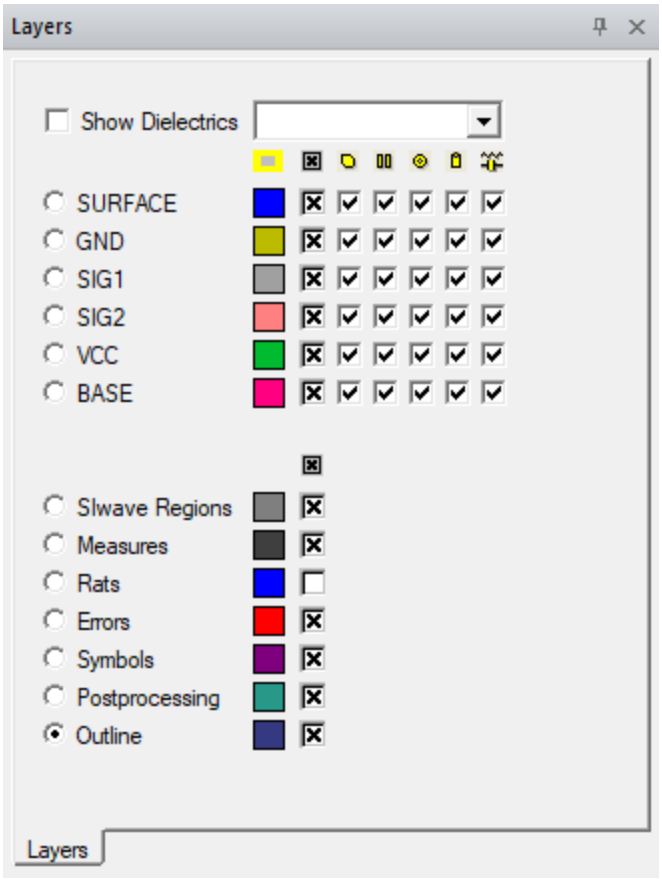
3D Layout offers several ways to create and configure layers:

- The Edit Layers window is the most comprehensive control.
- The Layers window has controls for layer visibility and select the active layer for a project in the **Layout Editor**.
- Controls on the **Layout** tab open the **Edit Layers** window and select the active layer.

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 the Layers Window

In the **Layers** window, you can select the working layer and the visibility of layout objects on specific layers, objects such as shapes, pads, components, nets, and the layers themselves. The **Layers** window is divided into three sections: stackup layers, non-stackup layers, and a preprocess mesh section. This dockable window can be resized and relocated.



- To show or hide the **Layers** window on the desktop, click **View** > **Layers**. A check box appears next to this command if the **Layers** window is visible.
- To select a working layer, click the radio button of the layer you want to work on.
- To view specific items on different layers, use **Show Dielectrics**, **Zone Filter**, the layers check boxes, and View Name drop-down menu.

If the stackup is a multizone stackup, select a zone from the **Zone Filter** to see the available layers.

The **Show Dielectrics** check box controls whether dielectric stackup layers are shown. If unchecked, dielectric layers are hidden and holes (vias and pins) are rendered continuously visible from the top-most visible signal layer to the bottom most visible layer. If checked,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dielectric layers appear and hole visibility can be finely controlled by each layer (but not necessarily continuously visible).

The View Name drop-down menu lists user-defined views and their controls. You can configure how layers and nets are displayed and then save this visibility as a view. Later, select that view from the drop-down menu to reset the **Layout** window to it. The view controls and default options are:

- **<Surface>** shows the top and bottom conducting layers of the layer stackup and hides all other layers. This is the default view.
- **Save Current View** opens a dialog that allows you to save the current setting of layers and nets to a new layout view.
- **Delete View** opens a dialog that allows you to delete saved layout views.

Check boxes control what is displayed in a layer. Icons at the top of a column control the entire column. Individual check boxes control the item visibility for the layer corresponding to the check box's row. Hover over an icon to see what the column controls:

Name	Icon	Controls the visibility of	Appears
Fill/Unfill All		Color in the object	Always
Show/Hide All		All objects	Always
Shapes		Shapes	Always
Lines		Lines and paths	Always
Pads		Pads	Always
Holes		Holes	Always
Components		Components	Always
Mesh		Mesh of signal layers	When a preprocess mesh is displayed
Background Dielectric Mesh		Mesh of dielectric layers	When a preprocess mesh is displayed with a laminate stackup

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A preprocess mesh section may appear below the non-stackup layer section. It controls design-wide visibility of:

- Airbox non-mesh and mesh
- Hole, solderball, and bondwire mesh

The visibility of these objects is not restricted to their layers. Also, layer visibility check boxes cannot control these mesh visibilities. Non-mesh visibilities are controlled by the layer check boxes.

Activating Layers

To draw on a layer, it must first be activated. This can be done on the **Layers** workspace, the **Edit Layers** window, or the **Layout** tab. Follow the steps below to activate a layer from each location.

Layers Workspace

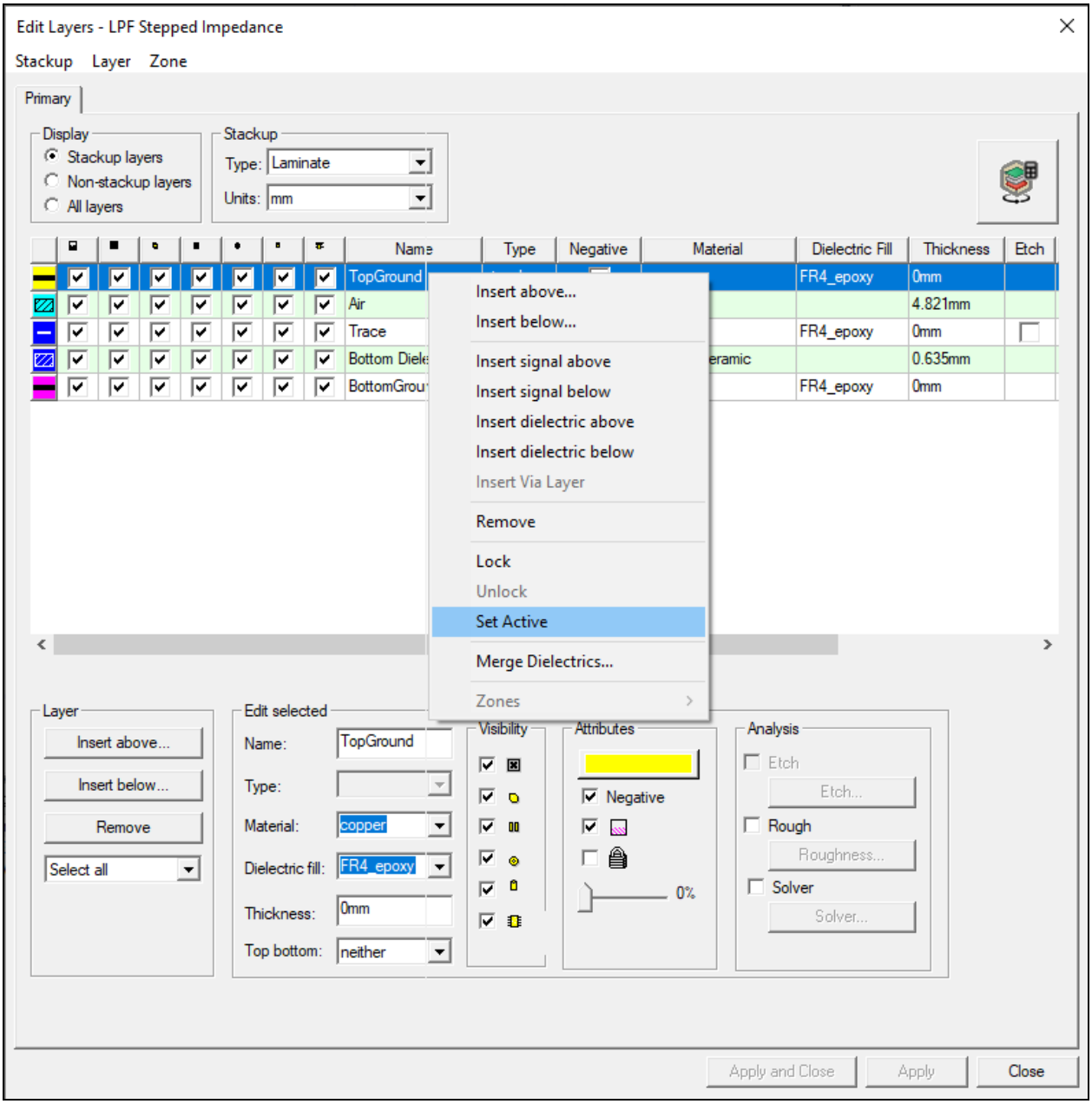
To open the **Layers** workspace, navigate to **View** > **Layers**, or click the **Layers** tab on the right-hand side of the **Layout Editor**.

From the **Layers Workspace**, select a layer to activate it.

Edit Layers Window

1. From the **Layout Editor**, perform one of these steps to open the **Edit Layers** window.
    a. From the **Layout** tab, select the **Layers dialog** icon.
    b. Navigate to the **Layout** > **Layers** .
2. From the **Edit Layers** window, right-click an inactive layer and select **Set Active**.
3. Click **Apply and Close** to close the **Edit Layers** window.

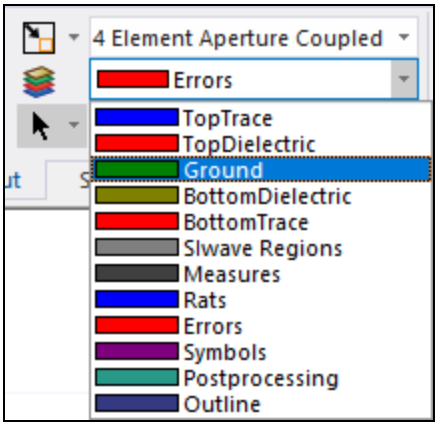
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Layout Editor Ribbon

From the **Layout Editor**, select an inactive layer on the **Layers dialog** drop-down menu to activate the inactive layer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The procedure is complete.

Adding and Removing Layers

Layers can be added three ways: though the **Layer** menu, by right-clicking in the **Stackup** page, and in the **Layer** pane. With each method, select whether to insert the new layer above or below a layer selected in the **Grid Control** pane. Right-click **Grid Control** to access quick options to insert a new signal or dielectric layer.

Via layers are only available in overlapping stackups and are added in a unique process.

Adding a Layer

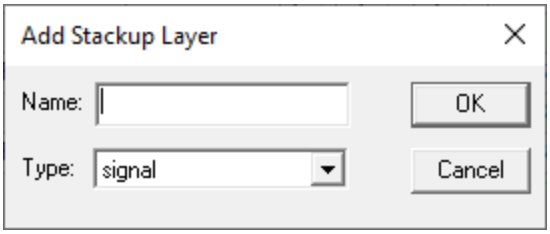
1. From the **Grid Control** pane of the Edit Layers window, click a layer that are above or below the new layer.
2. Do one of the following:
    - From the **Grid Control** pane, right-click selected layer and click an **Insert** command.

      > **Note:**
      >
      > Clicking **Insert signal** or **Insert dielectric** command adds the corresponding layer with no further steps.

    - Open the **Layer** menu and click an **Insert** command.

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 **Layer** pane, click **Insert** to open the **Add Stackup Layer** window.



3. Enter a name for the new layer in the **Name** field.
4. Select a type of layer on the **Type** drop-down menu. Different types are available for physical (stackup) layers and non-physical (non-stackup) layers.

Type	Physical Layer	Purpose
signal	Y	For drawing traces.
dielectric	Y	Acts as a substrate upon which signal layers border. Infinite in the XY plane.
assembly	N	For assembly geometry and text. Only one assembly type layer can be defined.
silkscreen	N	For silkscreen geometry and text. Only one silkscreen type layer can be defined.
soldermask	N	For soldermask geometry. Only one soldermask type layer can be defined.
solderpaste	N	For solderpaste geometry. Only one solderpaste type layer can be defined.
glue	N	For glue geometry. Only one glue type layer can be defined.
user	N	For user-defined geometry and text. Any number of user type layers can be defined.
post-processing	N	Provides placement for a 3D Line primitive that can then be used for post-processing calculations.

5. Click **OK**. The layer is added above or below the selected layer, depending on what was picked.

Removing a Layer

1. From the **Edit Layers** window, click layer(s) to remove them from the stackup.
2. Do one of the following:
   - From the **Grid Control** pane, right-click unwanted layer and click **Remove**.
   - Open the **Layer** menu and click **Remove** command.

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 **Layer** pane, click **Remove**.

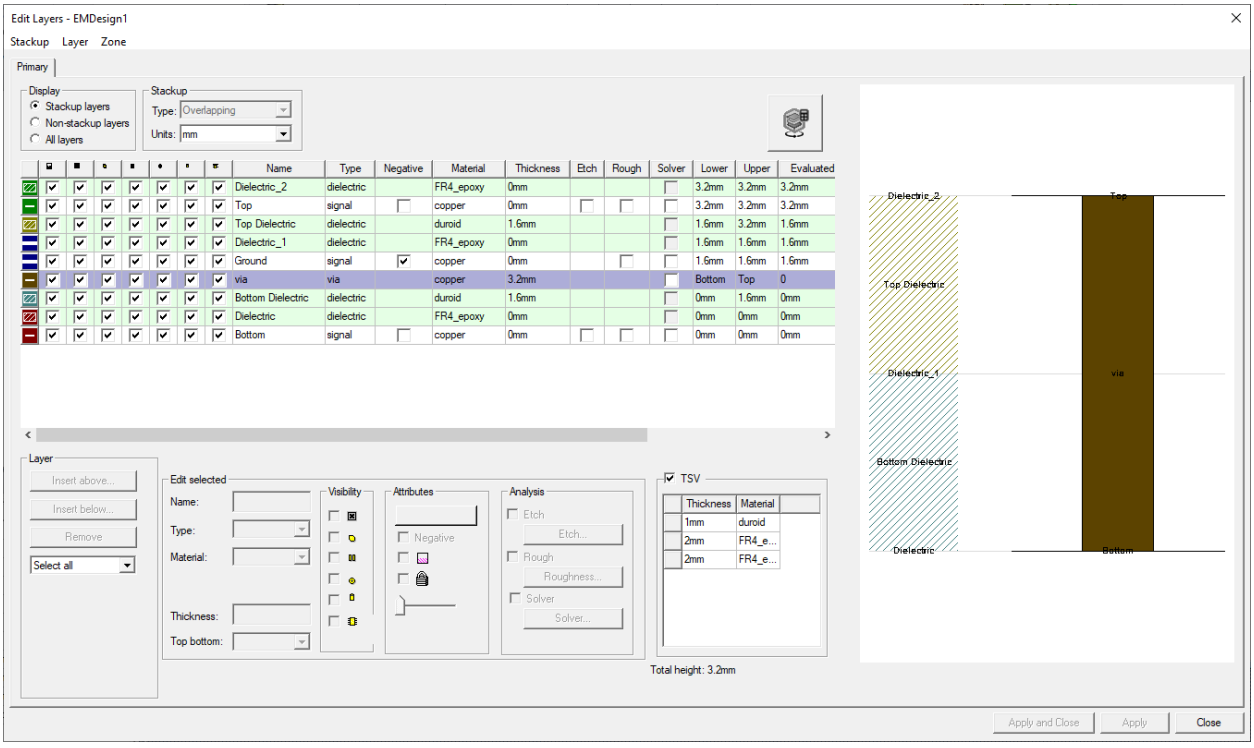
3. Confirm the action. The selected layers are removed on the stackup.

Working With TSV Layers

TSV (Through Silicon Via) layers are vias that can pass through semi-conductor layers, such as silicon. TSV layers require oxide layers along the via barrel to prevent any electrical contact between the barrel and the semi-conductor layers through which they pass.

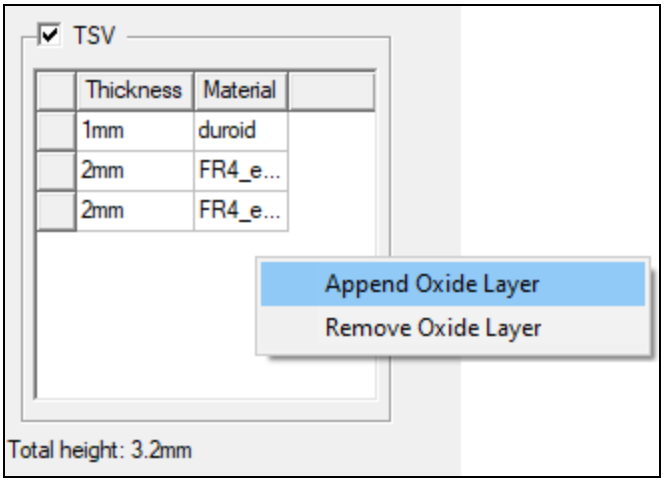
Once a via layer is created (refer to **Via Layers**), add, remove, or configure oxide layers by following these steps.

1. Open the **Edit Layers** window, by performing one of these steps on the **Layout Editor**:
   a. From the **Layout** tab, select the "Layers" icon.
   b. From the **Layout/Footprint** menu, select **Layers** .
   c. Right-click in the **Layout Editor** and select **Edit Layers**.
2. From the **Edit Layers** window, make the **TSV** area under the **Grid Control Table** by selecting a via layer.

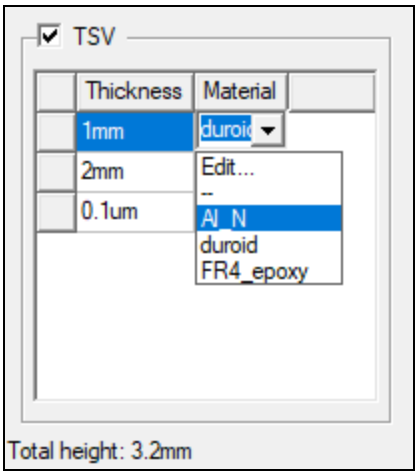


3. Right-click in the **TSV** area and select **Append Oxide Layer** (to add an oxide layer) or right-click the box adjacent to a layer and select **Remove Oxide Layer**, as appropriate.

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 in a **Thickness** field to adjust the width of an oxide layer, or select a different material makeup for the oxide layer on the adjacent drop-down menus, as appropriate.

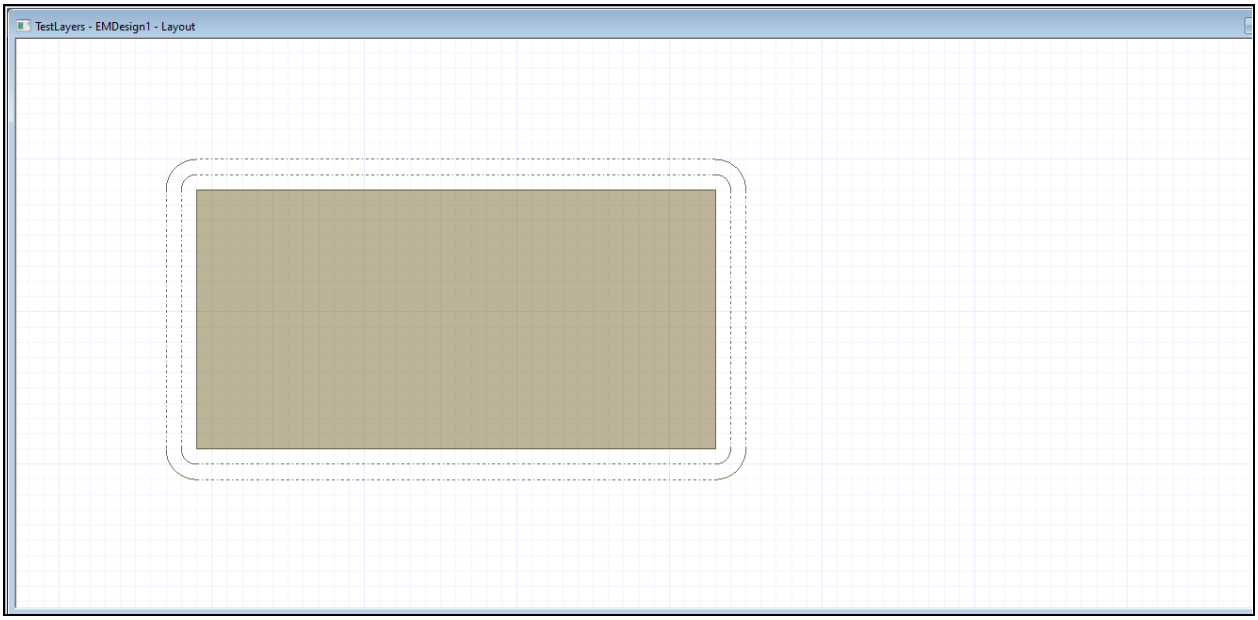


> **Note:**
>
> Make further modifications to the oxide layer materials by selecting **Edit** from the drop-down menu, to open the **Select Definition** window. Refer to **Assigning Materials**.

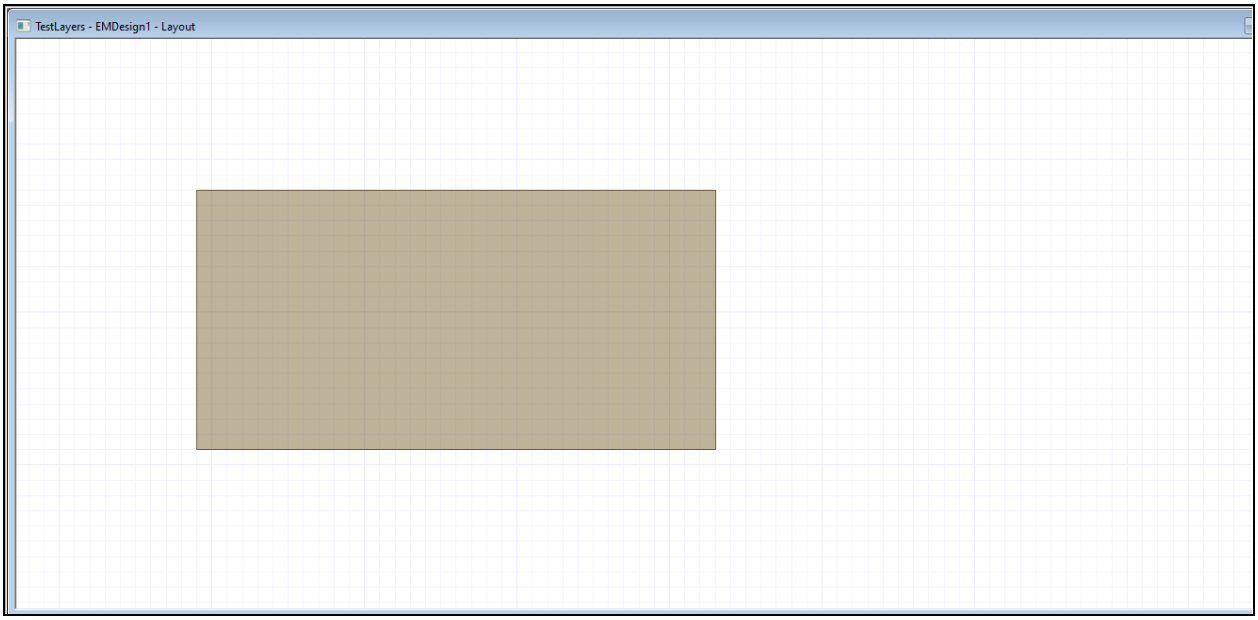
The procedure is complete. Refer to the following screenshots to see how oxide layers are depicted in the **Layout Editor**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active Design With Oxide Layers



Active Design Without Oxide Layers



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Materials to Layers

From the Edit Layers window, click the material of a layer in a drop-down menu on the **Primary** tab, and on the **Materials** tab of the **Select Definition** window. Materials can also be parameterized with project variable text array references.

Follow these steps to assign a material to a layer.

1. From the **Edit Layers** window, click rows to select the layers that you want to modify.
2. From the **Material** drop-down menu of the **Edit selected** pane, select a material or select **Edit** to open the **Select Definition** window.
3. If you chose **Edit**, select the material you want and click **OK** to close the **Select Definition** window.
4. Click **Apply** and the selected material is assigned to the selected layer.

Default Layers

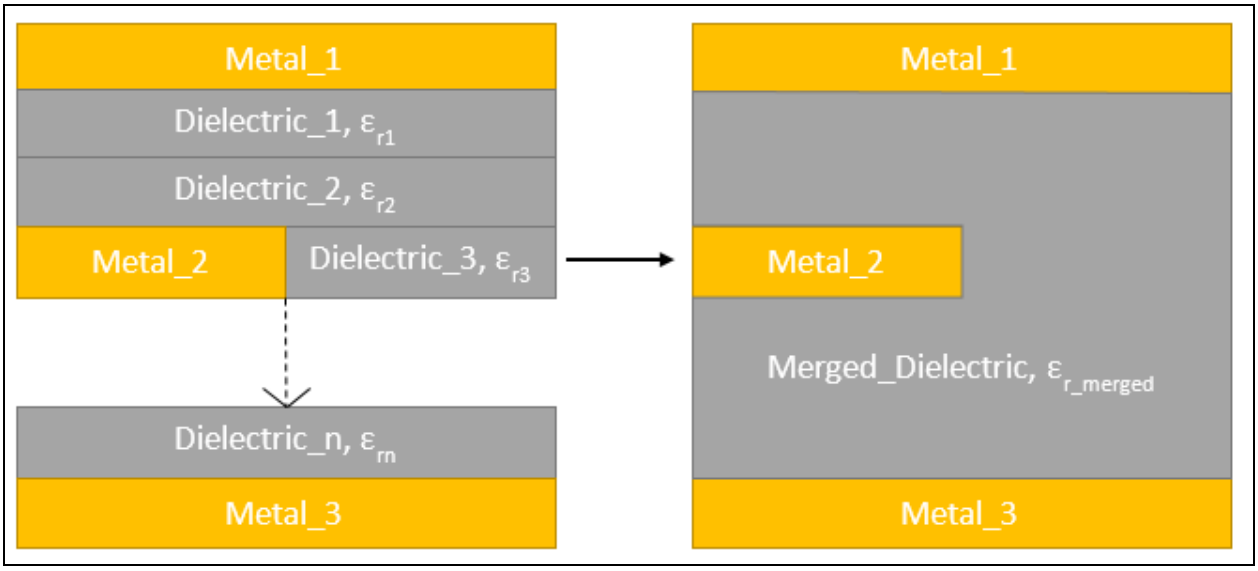
The following default layers are provided by HFSS.

Errors	Contains error messages generated during project setup.
Measures	Contains all measurements made using the ruler.
Outline	Contains the outlines of the board, its zones, and primitives.
Post-processing	Provides placement for a 3D line primitive that can then be used for post-processing calculations.
Rats	Contains logical connections between different components/circuit elements.
SI Regions	Created with an HFSS solution setup of the SIwave SYZ Solver.
Symbols	Contains the circuit element symbols.

Dielectric Simplification (Merging)

Dielectric simplification allows you to select one or more groups of continuous dielectric layers, then merge each group into one dielectric layer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



During this process, a relative permittivity for the new merged layer is calculated using one of the three methods in the following table. Select the appropriate method viathe **Merge Dielectrics** window.

Method	Equation
Weighted Capacitance	$\varepsilon_{r_merged} = \frac{\sum_{i=1}^{n} h_i}{\sum_{i=1}^{n} \frac{h_i}{\varepsilon_i}}$
Kraszewski Method	$\varepsilon_{r_merged} = \left(\frac{\sum_{i=1}^{n} h_i \sqrt{\varepsilon_i}}{\sum_{i=1}^{n} h_i} \right)^2$.
Weighted Average	$\varepsilon_{r_merged} = \frac{\sum_{i=1}^{n} h_i \varepsilon_i}{\sum_{i=1}^{n} h_i}$

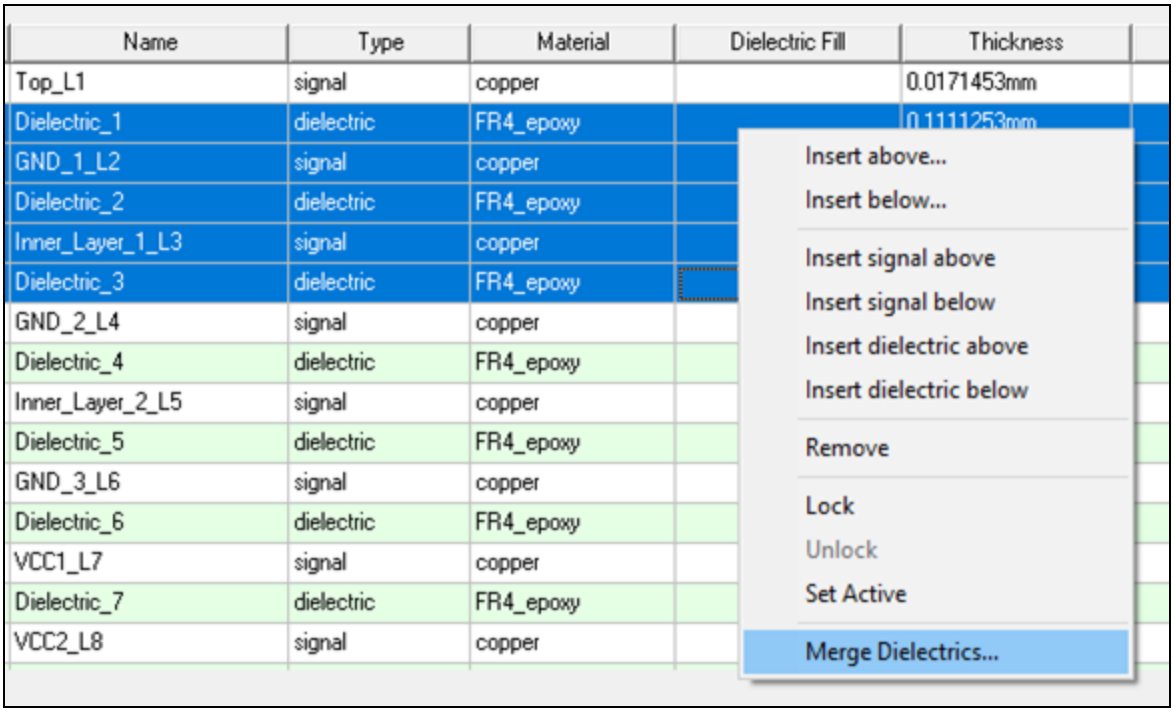
By combining numerous thin dielectric layers into one layer, the number of meshing elements is reduced with minimal decrease in computation accuracy. Thus, a comparable analysis can be performed in less time. Dielectric merging can be performed through either the user interface or the EDB API.

To perform dielectric merging:

1. Open the **Edit Layers** window.
2. Select one or more groups of layers.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

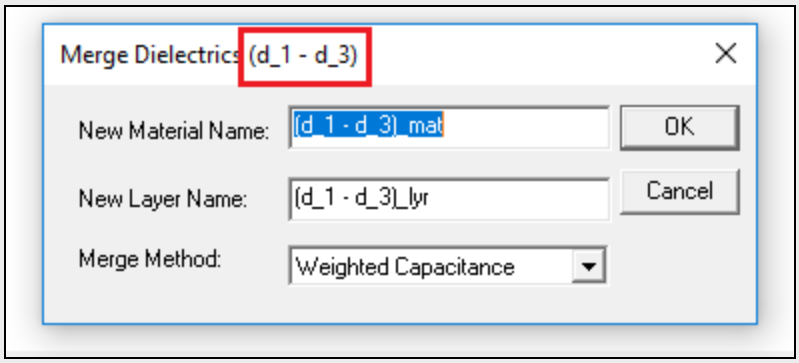
3. Right-click on a group of layers, then select **Merge Dielectrics**.



4. Enter the selected parameters in the **Merge Dielectrics** window.

Note:

When multiple dielectric ranges are selected, multiple **Merge Dielectrics** windows appear. The window titles include the range of layers affected that window's merge operation.



5. Click **OK** to merge the dielectrics.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drawing on a Layer

You can draw on one layer at a time in the **Layout Editor**. Select the layer you want to draw upon on the **Active Layer** drop-down menu on the **Layout** toolbar and begin drawing. Any objects drawn are on your selected layer.

Setting Layer Associations

To match two similar, but not identical designs, specify layer associations. The outcome of the matched layers depends upon which layer of a structure (top or bottom) you add the design.

The **Top Bottom Association** column indicates layer associations. From the Edit Layers window, the **Top Bottom Association** column is not visible by default. Right-click the grid control column headers and select **Top Bottom Association** to make the column visible. Alternately, select a row and choose an association on the **Top bottom** field in the **Edit Selected** pane. Then choose one of the following from the drop-down menu:

Item	Description
Top	The layer is associated with the top surface. For example, if a subcircuit is added to the top layer of a stackup, the subcircuit's **top** -associated layers are mapped to the stackup's **top** -associated layers. The subcircuit's **neither** -associated layers are subsequently mapped to layers of the same type (e.g., signal to signal) beginning at the top and moving downward. If a subcircuit is added to the bottom layer of a stackup, this mapping is reversed; **top** -associated layers are mapped to **bottom** -associated layers and **neither** -associated layers are mapped to like layers, beginning at the bottom and moving upward.
Bottom	The layer is associated with the bottom surface. For example, if a subcircuit is added to the top layer of a stackup, the subcircuit's **bottom** -associated layers are mapped to the stackup's **bottom** -associated layers. The subcircuit's **neither** -associated layers are subsequently mapped to layers of the same type (e.g., signal to signal) beginning at the top and moving downward. If a subcircuit is added to the bottom layer of a stackup, this mapping is reversed; **bottom** -associated layers are mapped to **top** -associated layers. The subcircuit's **neither** -associated layers are subsequently mapped to layers of the same type, beginning at the bottom and moving upward.
Neither	The layer is not associated with a top or bottom surface. **Neither** -associated layers are mapped to layers of the same type. For example, if a subcircuit is added to the top layer of a stackup, the subcircuit's **neither** -associated layers are mapped to layers of the same type (e.g., signal to signal) beginning at the top and moving downward. If a subcircuit is added to the bottom layer of a stackup, this mapping is reversed; **neither** -associated layers are mapped to like layers, beginning at the bottom and moving upward.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Layer Order and Elevations

Layer order is the sequence of physical layers in a stackup along the Z-axis. The order of the layers and their physical thickness determine the elevation of each layer.

Elevation in a stackup is the distance from zero (0) on the Z-axis to a point in a specified layer in a design's global coordinate system. A layer's lower elevation is the lowest point of the layer, while its upper elevation is the highest point of the layer. Depending on the type of stackup, change a layer's elevation without changing the layer's position in the stackup.

Elevation in Laminate and Multizone Stackups

In laminate and multizone stackups, each layer lays on the layer underneath it. Signal layers cannot be embedded in dielectric layers. The thickness of each layer can be changed but not its lower elevation. Because of this, the only ways to change a layer's lower elevation in a laminate or multizone stackup is to:

- Add or remove a layer
- Change the order of the layer
- Change the thickness of a layer underneath it.

Elevation in Overlapping Stackups

In overlapping stackups, dielectric layers must be continuous (i.e., stack on top of one another). They cannot overlap and there can be no gaps between them. To create an air gap between two dielectric layers, you must explicitly add a dielectric layer between them and define its material as "air".

In contrast, signal layers can reside at arbitrary elevations and be embedded in a dielectric layer or at the interface of layers. Signal layers and negative signal layers can overlap one another. They can also have non-zero thicknesses.

Via layers must connect two signal layers.

To help you understand where each layer in an overlapping stackup is located, four elevation columns appear by default for overlapping stackups. The **Lower** and **Upper** columns refer to the layer's elevations and can contain variables or expressions. The **Evaluated Lower** and **Evaluated Upper** columns provide calculated values for their elevations.

To change the order of layers in an overlapping stackup, choose on the following instructions.

Method A

1. From the Edit Layers ' control grid, change the **Lower** value of the bottom most dielectric layer. This is the only layer whose **Lower** value can change.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Change the **Thickness** and **Material** of the other dielectric layers as selected.
3. Add and remove dielectric layers as appropriate.
4. Change the **Lower** value for each signal layer that should move.

Method B

1. Change the stackup's **Type** to a **Laminate**.
2. **Click**+**drag** the dielectric layers into the correct order.
3. Change the stackup's **Type** back to **Overlapping**.
4. Changing the **Lower** value for each signal layer.

Setting the Layer Lower Elevation

The lower elevation of a layer is the lowest point of the layer (e.g., if you create a layer with a lower elevation of 2 microns, the layer begins at 2 and extends upwards until it reaches the **Upper Elevation** value). A signal layer's Lower Elevation value indicates its elevation above zero. Lower elevation can only be changed on signal layers in non laminate stackups.

To set a layer's lower elevation:

1. From the Edit Layers window, in the **Lower Elevation** column, double-click a layer's **Lower Elevation** field.
2. Type a value and its units in the field. The value can be specified using a number (e.g., 0.5um), a variable (e.g., $H), or an expression (e.g., $H-0.5um+$W).
3. Press **Enter**. The layer is assigned the lower elevation you entered.

Setting the Layer Upper Elevation

The upper elevation of a layer is the highest point of the layer. The upper elevation of a layer adjusts itself based on the lower elevation and the thickness of the layer. The Upper Elevation is not directly editable, but it can be seen in the Edit Layers window, in the **Upper Elevation** column.

Setting Layer Thickness

The thickness of each layer is listed in the **Thickness** column of the **Grid Control** pane in the **Edit Layers** window. A layer's thickness is measured on the point of elevation (e.g., if the elevation of a signal layer is 2 microns and the thickness is 4 microns, the layer begins at 2 microns and extends upwards for 4 microns to a height of 6). The units for the thickness is in the **Units** field of the **Stackup** pane.

Follow these steps to set a layer's thickness.

1. From the **Thickness** column of the **Edit Layers** window, double-click a layer's **Thickness** field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

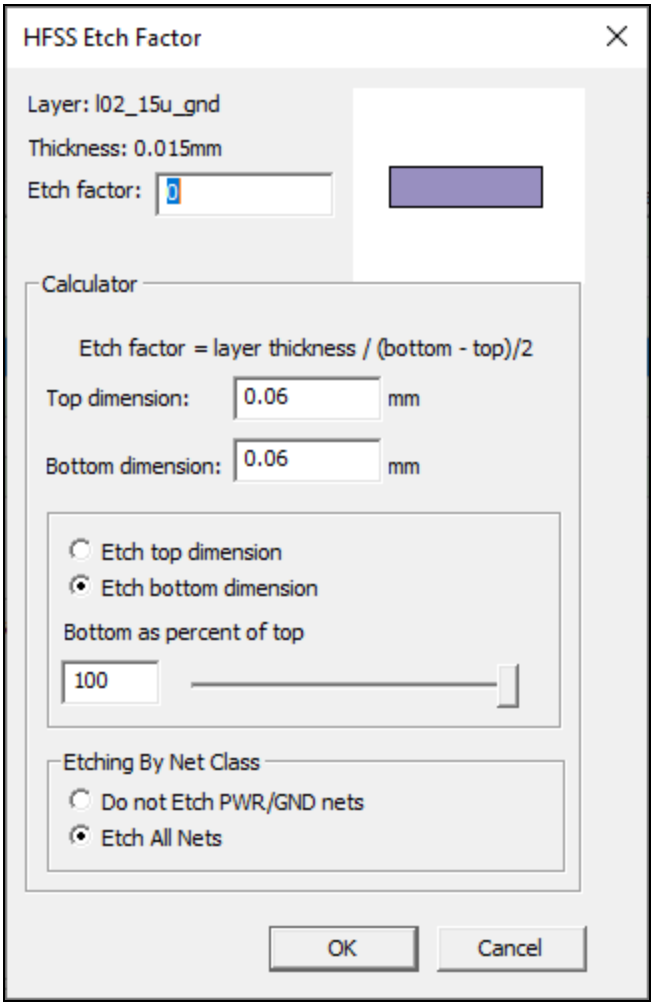
2. Type a value and its units in the field. The value can be specified using a number (e.g., 0.5um), a variable (e.g., $H), or an expression (e.g., $H-0.5um+$W).
3. Press **Enter**.The layer is assigned its new thickness.

> **Warning:**
>
> Assigning thickness to signal layers significantly increases the solution time. The number of unknowns that HFSS software must compute doubles when you increase the thickness from zero (infinitely thin) to a finite number. Therefore, if possible, model signal layers as infinitely thin objects.

Setting the Etch Factor

Substrate manufacturing does not create exact geometry. The actual geometry of substrate manufacturing often results in different thicknesses in the dimensions of each geometry within a layer — and that difference is represented by an “etch” or “etch factor”. HFSS geometry allows the presence of an etch factor in the design and modeling of different substrate layers. The specific etch factor that applies to a particular layer is usually available from the manufacturer, but is generally calculated by taking the ratio of etch depth to the undercut, where the undercut is defined as being the part of the wafer that is etched away under the photoresist. The etch factor is configurable on the **HFSS Etch Factor** window.

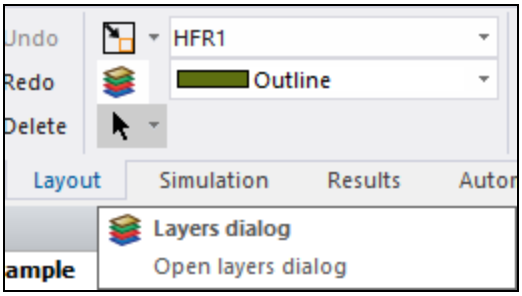
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Accessing the HFSS Etch Factor Window

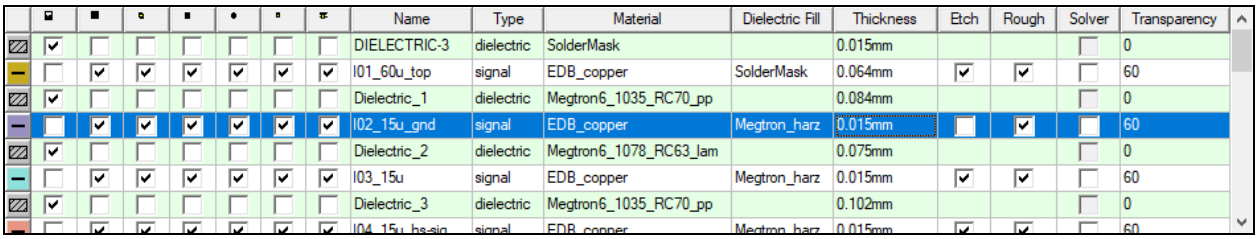
Follow these steps to access the **HFSS Etch Factor** window.

1. From the **Layout** tab, click the **Layers dialog** button to open the **Edit Layers** window.



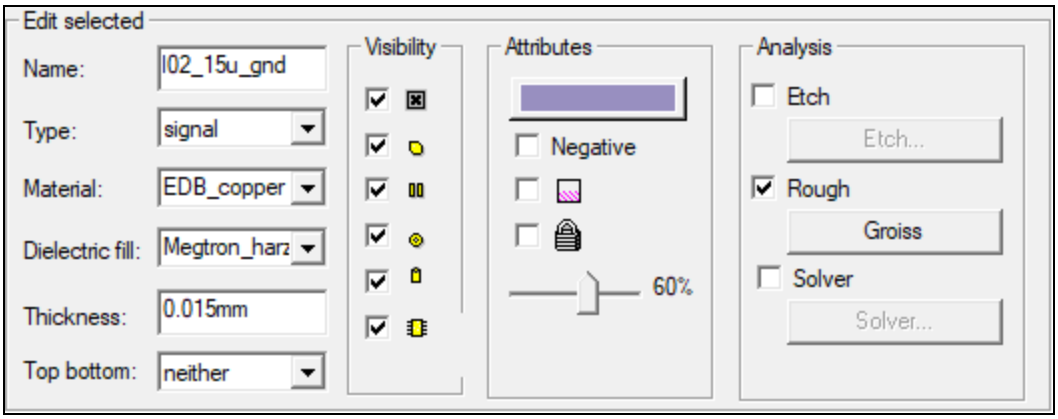
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select a row in the layer table that has a check box in the **Etch** column.

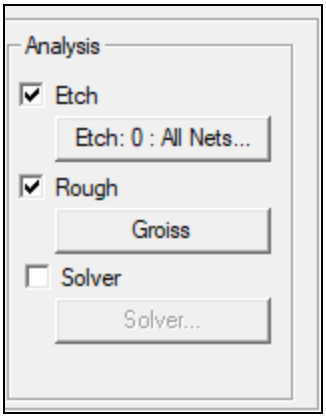


	Name	Type	Material	Dielectric Fill	Thickness	Etch	Rough	Solver	Transparency
	DIELECTRIC-3	dielectric	SolderMask		0.015mm				0
	l01_60u_top	signal	EDB_copper	SolderMask	0.064mm	☑	☑		60
	Dielectric_1	dielectric	Megtron6_1035_RC70_pp		0.084mm				0
	l02_15u_gnd	signal	EDB_copper	Megtron_harz	0.015mm		☑		60
	Dielectric_2	dielectric	Megtron6_1078_RC63_lam		0.075mm				0
	l03_15u	signal	EDB_copper	Megtron_harz	0.015mm	☑	☑		60
	Dielectric_3	dielectric	Megtron6_1035_RC70_pp		0.102mm				0
	l04_15u_hs-sig	signal	EDB_copper	Megtron_harz	0.015mm	☑	☑		60

3. Navigate to the **Edit selected** area > **Analysis** area.

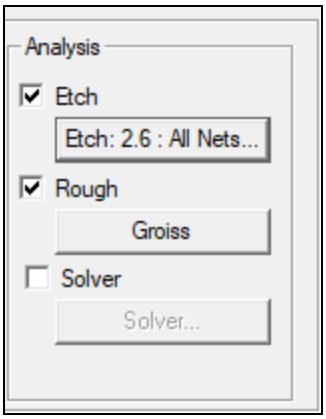


Ensure the **Etch** box is checked to activate the button beneath it.



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 button to open the **HFSS Etch Factor** window.

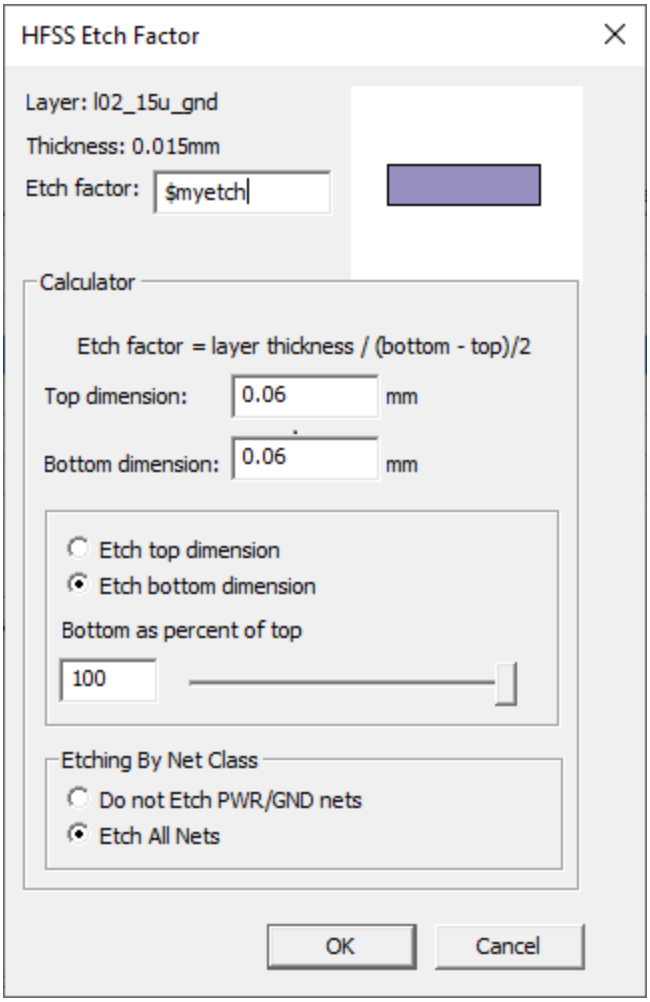


5. Make any adjustments as selected. Then click **OK** to close the window and apply the etch factor parameters to the selected layer.

Entering a New Etch Factor

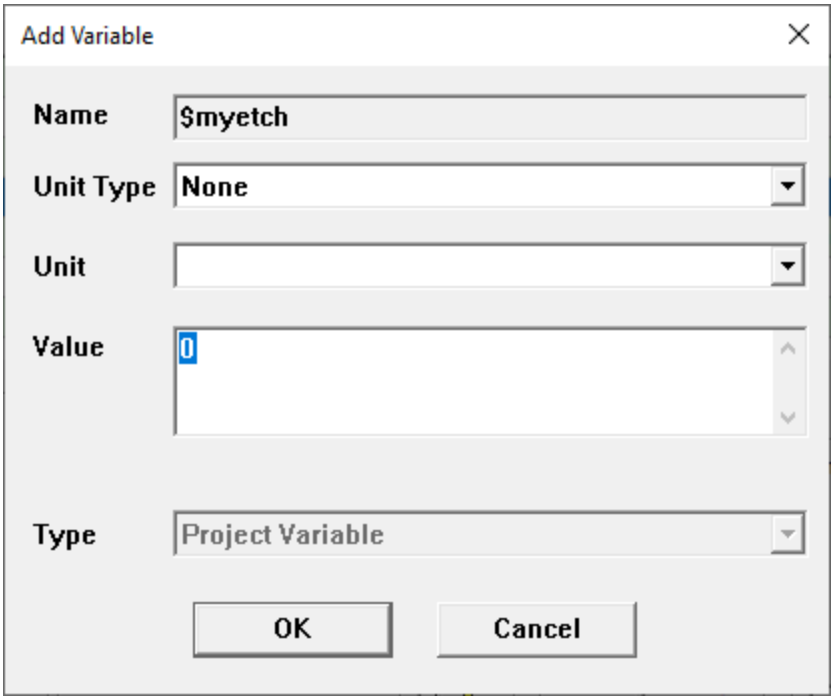
The **Etch factor** value may be entered as a number or may be parameterized as a project variable.

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 variable has not yet been created, an **Add Variable** window will appear when users navigate away on the **Etch factor** field. Refer to **"Working with Variables" on page 3-104**.

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 input to that control is a number, the calculator controls are available; otherwise, the calculator controls are deactivated.

The Calculator Area

The purpose of the etch factor calculator is to give you a way to calculate an etch factor based on layer thickness. By default, the **Top** and **Bottom** dimension fields are populated with values that correspond to the value in the **Etch factor** field. However, the values are representational and do not correspond to a particular geometry in the layout.

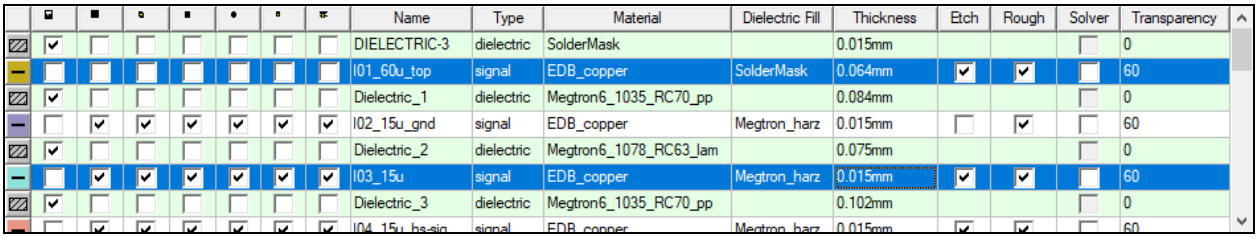
To alter the etch factor, either enter new values in the **Top** and **Bottom** dimension fields, or enter a percentage into the **Top/Bottom as percent of top** field directly or by adjusting the slider to set the value of one dimension to a percentage of the other. As the **Top** and **Bottom** dimensions change, the etch factor and subsequently the layer drawing are modified to reflect the change.

The Etching By Net Class Area

By default, **Etch All Nets** is selected. Select **Do not Etch PWR/GND nets**, and any power/ground nets in the selected layer will **not** be etched.

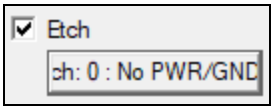
Change the **Etch All Nets**/**Do not Etch PWR/GND nets** option for multiple layers simultaneously by using **Ctrl**+**click** to highlight multiple rows in the layer table.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	Type	Material	Dielectric Fill	Thickness	Etch	Rough	Solver	Transparency
DIELECTRIC-3	dielectric	SolderMask		0.015mm				0
l01_60u_top	signal	EDB_copper	SolderMask	0.064mm	☑	☑		60
Dielectric_1	dielectric	Megtron6_1035_RC70_pp		0.084mm				0
l02_15u_gnd	signal	EDB_copper	Megtron_harz	0.015mm		☑		60
Dielectric_2	dielectric	Megtron6_1078_RC63_lam		0.075mm				0
l03_15u	signal	EDB_copper	Megtron_harz	0.015mm	☑	☑		60
Dielectric_3	dielectric	Megtron6_1035_RC70_pp		0.102mm				0
l04_15u_hs-sig	signal	EDB_copper	Megtron_harz	0.015mm	☑	☑		60

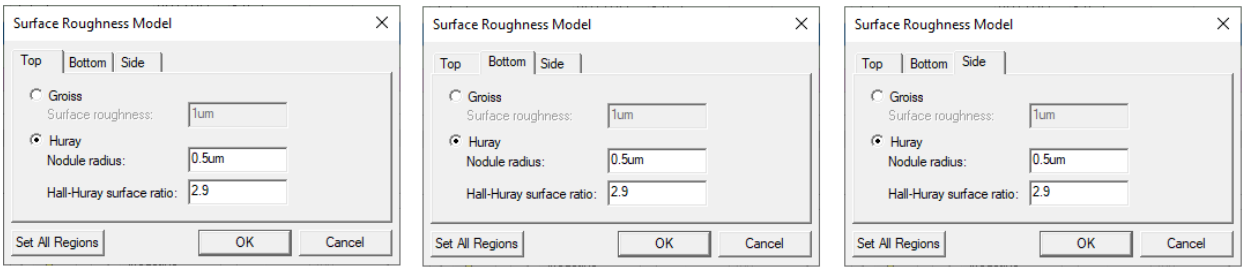
The choice is also reflected on the **Edit Layers** window > **Etch** button.



Setting the Layer Surface Roughness

The surface roughness model is used to compute conduction losses for surfaces. Surface roughness increases conduction losses, such as the interface between the conductor and the substrate for a microstrip line. The **Surface Roughness Model** window is accessed through the Edit Layers window. To access the Surface Roughness window:

1. Check the **Rough** area for a selected layer in the **Edit Layers** window.
2. Click **Roughness** to open the **Surface Roughness Model** window.



The surface roughness of top and bottom layers is set independently. Also set one roughness for the sides. There is a separate tab in the window for each.

3. Select either the **Top**, **Bottom**, or **Side** tab.
4. Select either **Groiss** or **Huray:**
   a. For the Groiss model, specify a **Surface roughness** parameter as a value/variable, then specify the corresponding units. The Groiss model is a traditional-case model. It is not causal and is only suitable for frequency-domain computations. It is also limited to a maximum impedance multiplying factor of 2, which corresponds to highly polished conductor surfaces. Legacy projects use the Groiss model by

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default.

b. For the Huray model, also specify the **Nodule radius** value, which is the radius of copper spheres used to model the surface roughness. Next, specify the **Hall-Huray surface ratio**, which is a unitless quantity. The Huray model is causal.

5. The surface roughness of each surface is set independently. To copy the settings from one surface to the other two:
   a. Set the selected model for one of the surfaces.
   b. Click **Set All Regions**.
6. Click **OK**.

Groiss Surface Roughness Model

The Groiss surface roughness model is a simple function of the surface roughness and skin depth of the conductor.

The smooth surface impedance (Z_{smooth}) is increased by a factor ($fact_R$), given by the equation (1):

$$fact_R = 1 + e^{-\left(\frac{\delta}{2s}\right)^{1.6}}$$

where:

- s is the surface roughness (in length units)
- δ is the skin depth, which is defined as (2):

  $$\delta = \frac{1}{\sqrt{\pi f \mu_r \mu_0 \sigma}}$$

  in which:

- f is the frequency in Hz
- μ_r is the relative permeability of the conductor
- μ_0 is the permeability of free space

  $$\mu_0 = 4\pi \cdot 10^{-7}\ \mathrm{H/m}$$

- σ is the electrical conductivity in Siemens/m

Once the Groiss surface roughness factor is determined, the rough surface impedance (Z_{rough}) is determined as (3):

$$Z_{rough} = fact_R \cdot Z_{smooth}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limitations of the Groiss Model:

The Groiss surface roughness model is non-causal and is therefore limited to the frequency-domain (not suitable for time-domain computations). The Groiss roughness factor is bound by the following limits:

$0 < fact_R \leq 2$

Therefore, it is suitable for relatively highly polished conductors and not suitable for the much rougher copper conducting traces that are typical of manufactured PCBs.

For roughness factors greater than 2 and for time-domain computations, use the Huray Surface Roughness Model.

References:

1. S. Groiss, I. Bardi, O.Biro, K. Preis and K.R. Richter, Graz University of Technology, Kopemikusgasse 24, A-8010 Graz, Austria, "Parameters of Lossy Cavity Resonators Calculated by the Finite Element Method", IEEE TRANSACTIONS ON MAGNETICS, VOL 32. NO 3, MAY 1996

Huray Surface Roughness Model

Surface roughness can increase conductor power losses by more than two times the loss associated with smooth conductors. In Ansys Electronics Desktop, a modification of the standard real-valued Huray surface roughness model is implemented to accurately account for these losses in both the frequency and time domains.

This modification fixes a causality problem with the original Huray model by adding some frequency-dependent reactance to the roughness-enhanced surface impedance. The purpose is to provide a complex-valued causal analytic model that exactly matches the loss results of the original model but is also suitable for time domain computations.

The original Huray model was limited to calculations in the frequency domain and did not account for the effects of roughness on the phase of the transmitted signal. These limitations have been eliminated. The enhanced causal model not only provides accurate broadband modeling of the losses in very rough copper foils (typically used in PCB manufacturing) but also accurately predicts increased phase delay due to the surface roughness.

Essentially, the smooth surface impedance (Z_{smooth}) is increased by a complex “Huray” factor (H_c), a function of frequency, given by the following equation (1):

$$H_c\left(j\omega\right) = 1 + \frac{K}{1+\left(\frac{j2\omega}{\omega_0}\right)^{-1/2}}$$

where:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ω is the frequency in radians/second
- ω_0 is a corner frequency represented by the equation (2):

$$\omega_0 = \frac{2}{a^2 \mu_r \mu_0 \sigma}$$

- K is a derived constant used to simplify the equation (3):

$$K = \frac{3}{2} \cdot SR = \frac{6\pi a^2 N}{A_f}$$

in which:

- *SR* is the Hall-Huray surface ratio:

$$\frac{4\pi a^2 N}{A_f}$$

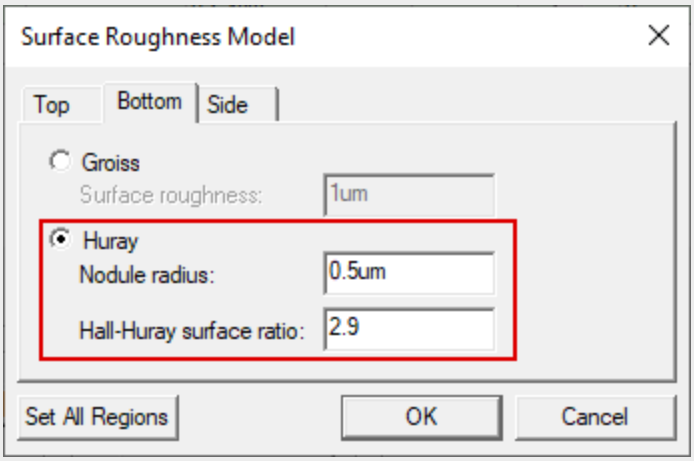
- *a* is the radius of a spherical nodule
- μ_r is the relative permeability of the conductor
- μ_0 is the permeability of free space

$$\mu_0 = 4\pi \cdot 10^{-7}\ \mathrm{H/m}$$

- σ is the electrical conductivity in Siemens/m
- *N* is the number of nodules
- A_f is a unit cell area

Note:

a and *SR* are the two parameters you must enter in the **Surface Roughness Model** dialog box when you select the Huray option:



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 about this complex causal Huray model (its necessity, background, derivation, proof of accuracy, and consistency with the original real-valued model), see Reference 6 below.

Once the Huray surface roughness factor is determined, the rough surface impedance (Z_{rough}) is determined by the equation (4):

$$Z_{rough} = H_c \cdot Z_{smooth}$$

References:

1. P.G Huray, S.G Pytel, S.H Hall, F. Oluwafemi, R.I. Mellitz, D. Hua, and P. Ye, "Fundamentals of a 3-D "Snowball" Model for Surface Roughness Power Losses", 11th Annual IEEE SPI Proceedings, May 13 – 16, 2007.
2. S.H Hall, S.G. Pytel, P.G Huray, D. Hua, A. Moonshiram, G. Brist, and E. Sijercic, "Multi-GHz, Causal Transmission Line Modeling Methodology with a Hemispherical Surface Roughness Approach", IEEE Transactions on Microwave Theory and Techniques, December 2007 pp 2614 – 2624.
3. S.G Pytel, P.G Huray, A. Moonshiram, S.H Hall, R.I Mellitz, G. Brist, F. Oluwafemi, H.M Meyer, L. Walker, and M. Garland, "Analysis of Copper Treatments and the Effects on Signal Propagation", 58th Annual IEEE ECTC, May 26 – 30, 2008, pp 1144 – 1149.
4. S.G Pytel, "Multi-gigabit data signaling rates for PWBs including dielectric losses and effects of surface roughness", PhD. Dissertation, University of South Carolina, 2007.
5. P.G Huray, O. Oluwafemi, J. Loyer, E. Bogatin, and X. Ye; "Impact of Copper Surface Texture on Loss: A Model That Works", DesignCon 2010, February 1 - 4, 2010.
6. J. Eric Bracken, Ansys, Inc., "A Causal Huray Model for Surface Roughness", DesignCon 2012, January 2012.

Working with Dielectric and Negative Signal Layers

If you have multiple dielectric layers in the model, they must touch other dielectric layers or negative signal layers. This prevents “pockets” of air from developing. If you have a signal layer between two dielectric layers, the dielectric layers must touch. The signal layer may overlap the two dielectric layers; however, a signal layer may not touch a negative signal layer. You may create an air gap by defining a dielectric layer as air.

Via Layers

Via layers are used to create a connecting metal layer between two existing signal layers. They are abstract constructs that exist between and in reference to their signal layers. For instance, a via layer’s Evaluated Lower and Evaluated Upper values are defined by the Evaluated Lower and Evaluated Upper values of their signal layers. The thickness of a via layer is the distance between its reference signal layers. Via layers are rendered as purple in the Edit Layers window's Control Grid.

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 via layer:

1. From the **Edit Layers** window, make the stackup an overlapping stackup.
2. Use the mouse and **Ctrl** key to select two signal layers between which the via layer exists.
3. Right-click on one of the selected layers and click **Insert Via Layer**. The new layer appears in the Stackup Viewer and Control Grid.

Existing layers in an overlapping stackup can be changed to via layers by selecting a layer and changing the **Type** drop-down menu in the **Edit selected** pane.

Drawing a Model in the Layout Editor

All geometric models that you create are a collection of objects. These objects are drawn at the **Layout Editor**. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform allows you to draw on all three primitive layer types (i.e., dielectric, signal, and negative signal).

The Drawing Region

The drawing region is the rectangular area, displayed as a grid, in which you create the model. You can control the following aspects of the drawing region:

Setting the Drawing Region Size

Defining the size of the region in which you are interested helps to conserve computing resources. HFSS does not attempt to compute a solution outside the drawing region.

Drawing region settings can be changed for the local layout design, or for all layout designs. See The Display Tab under Layout Options for details on changing the drawing region settings

Fitting View Areas in the Drawing Region

If the drawing region is too large to fit in the viewing area of a **Layout Editor**, scroll bars automatically appear on the bottom and left side of the window. Use them to scroll across the drawing region. Alternatively, click **View > Fit Drawing** to rescale the viewing area of the window to display the entire drawing region.

Choosing Grid Settings

The grid displayed in the **Layout** window is a drawing aid that helps to visualize the location of objects. By default a cartesian (rectangular) grid is displayed in the active **Layout Editor**. The grid is centered at the origin of the local coordinate system (x = 0, y = 0). Grid spacing is set according to the current project's drawing units.

Grid settings can be changed for the local layout design, or for all layout designs. See The Display Tab under Layout Options for details on changing the grid settings

Choosing 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.

By default, the selection point and graphical objects are set to “snap to” a point on the grid when the cursor hovers over it. The coordinates of the selection point are used, rather than the exact location of the mouse.

Snap settings can be changed for the local layout design, or for all layout designs. See The Snapping Tab under Layout Options for details on changing the snap settings

Choosing Shadow Settings

When a sub-circuit is selected, it is drawn shadowed. The shadowing can be controlled through a property on the sub-circuit. Shadow Percent: a low number means less shadowing, a high number means more. A low number also means the sub-circuit are drawn with colors closer to the original intensity.

General Drawing Guidelines for the Layout Editor

There is no strict procedure to follow when creating a geometric model; these steps serve as general guidelines.

1. Select the layer you want to draw on.
2. Activate the **Layout** view window by clicking its window frame.
3. Click the **Draw** menu commands to create one- and two-dimensional objects. Treat each conductor or material in the structure as a separate object.
4. If applicable, select commands from **Draw** and **Layout** to modify the geometry that you created.
5. Periodically save the geometry.

Drawing Apertures on Negative Signal Planes

You can create apertures or holes in an infinite negative signal plane layer. Draw an object on the negative signal layer in the shape of the aperture. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform assumes any object drawn on a negative signal layer is an aperture and automatically assigns it the properties of air (if the stackup is non laminate) or properties of a dielectric fill material (if the stackup is laminate).

Zooming and Panning the Layout View

Complete the steps in the following subsections to magnify (zoom in) or shrink (zoom out) the contents in the view window, or on a rectangular area in the view window, and pan (scroll) the view in any direction.

> **Note:**
>
> While zooming and panning, the absolute size of the active design 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.

Zoom In on Window Contents

1. To magnify (zoom in on) the contents in the view window, do one of the following:
    a. From the **View** menu, select **Zoom In**.
    b. Right-click in the **Layout Editor**/**Schematic Editor** and select **Zoom In**.
    c. Press **Ctrl** + **+**.
2. Repeat until the selected magnification is achieved.

Zoom Out on Window Contents

1. To shrink (zoom out on) the contents in the view window, do one of the following.
    a. From the **View** menu, select **Zoom Out**.
    b. Right-click in the **Layout Editor**/**Schematic Editor** and select **Zoom Out**.
    c. Press **Ctrl** + **-**.
3. Repeat the operation until the selected magnification is achieved.

Zoom In on a Rectangular Area

1. To magnify a specific rectangular area in the view window, do one of the following:
    a. From the **View** menu, select **Zoom Area**.
    b. Right-click in the **Layout Editor**/**Schematic Editor** and select **Zoom Area**.
2. The cursor changes to a magnifying glass. **Click**+**drag** to draw a rectangle (or square) by selecting two diagonally opposite corners. This is the area where magnification is increased.
3. Release the mouse button to magnify the area. The cursor returns to normal.

Zoom In/Out Using the Mouse Wheel

a. Press **Shift** and spin the mouse wheel to zoom in/out at the current cursor position.

b. Press **Shift** + **Alt** and spin the mouse wheel to zoom in/out more precisely.

Restoring a Previous Zoom View

After executing a zoom in or zoom out operation, you can return to the previous magnification by selecting **Zoom Previous** on the **View** drop-down menu or on the shortcut menu in the layout window.

Panning the 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 pan (scroll) the view in any direction, hold down **Shift** while clicking the mouse. The view is attached to the cursor. While holding down the Shift key and left mouse button, move the cursor to pan the view in any direction. Release the mouse button to end the panning operation.

Cross-Probing Selected Elements

When both the **Layout Editor** and the Schematic editor are displaying the same design, select one or more component instances and interface ports in both editors. Select schematic elements or layout elements to cross-probe. Then use one of the following methods to effect the cross-selection:

- In either editor, press **Ctrl**+**K**
- From the **Schematic Editor**, select **Schematic** > **Cross-Probe Layout**
- From the Layout editor, select **Layout** > **Cross-Probe Layout**

The corresponding elements in the related editor are selected and that editor window are brought to the front. If you select only component instances or only interface ports, then cross-probing makes the primary selection in the first editor the primary selection in the related editor. Note that all previous selections in the related editor are cleared and only the cross-probed elements are selected when the command is invoked.

Note:

Changes to a design should be made on the editor that was used to create the design. For example, if a design was created using the **Schematic Editor**, it should not be modified on the layout editor. In some cases, attempts to change a schematic design on the **Layout Editor** results in a warning such as:

```
[warning] Port Port2, ID %2, was disconnected from net 98, but
it was not possible to unwire it. This must be done manually.
```

Similarly, a **Layout Editor** design can be viewed as a schematic using the cross-probing feature, but the schematic view should not be used to make changes to the design. Although an operation such as deleting a port in the schematic may successfully delete the cross-probed layout port, the operation cannot be undone in the **Layout Editor** after the schematic window has been closed (the undo attempt may cause an error).

Voids in Layout

For both copy/paste and copy to an EM design, if a void's owner is selected, all of its voids are copied and pasted as voids. Any voids that are not wanted can then be deleted after the paste.

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 void is selected but its owner is not, the void is copied as a primitive. If selected, the boolean subtract functionality can be used to notch or add a void, based on the overlap between the primitive and another selected primitive.

Directional Drawing in the Layout Editor

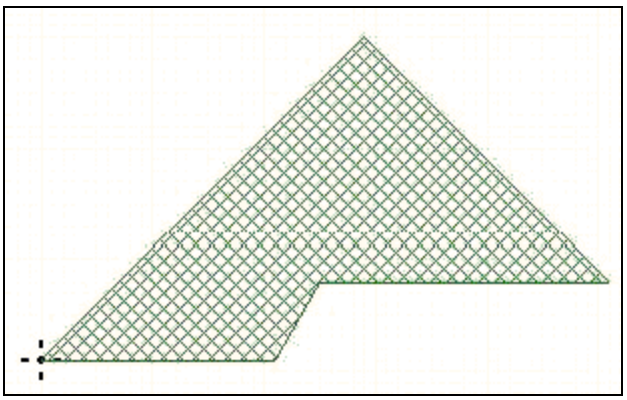
Directional drawing allows you to constrain snapping to a particular direction when moving, placing, and duplicating objects or primitives in the **Layout Editor**. Directional drawing/snapping can be set to snap to the orthogonal (x or y) or to 45 degrees. Directional drawing/snapping is set in the same manner as other layout snap options, by using the **Snapping** options tab of the **Layout Options** window. For more information see The Snapping Tab.

Rather than configuring a universal directional drawing constraint using the **Snapping** options tab of the **Layout Options** window, alternately you may temporarily modify the current constraint using a simple keystroke. The following keys modifies the current directional drawing/snapping constraint as you manipulate objects and primitives in the **Layout Editor**:

- Pressing 'd' applies a 45 degree directional drawing/snapping constraint
- Pressing 'r' applies an orthogonal directional drawing/snapping constraint
- Pressing 'n' removes any directional constraint that has been configured

Both the move and duplicate operations uses the currently configured directional constraint. The default for directional drawing is none. From the **Layout Editor**, the directional drawing mode is indicated by a dotted-cross cursor.

The reference point for directional drawing — the (0,0) point for the current movement — is marked with a dotted cross:



The following directional drawing rules apply when creating primitives, voids, and measurements:

- The first hover after initiating directional drawing sets the reference point for the current movement.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The cursor moves freely, but when a snapped-on-hover event occurs, the directional constraint is applied.
- Each click resets the reference point for the next movement.

Drawing Geometric Objects in the Layout Editor

You can draw one- or two-dimensional objects using **Draw** commands. You can alter objects individually or together to create the geometry of your structure.

- *One-dimensional (1D,)* or *open objects* are zero-width arcs, lines, or any combination thereof that have not yet been closed to form the boundary of an object. Generally, open objects are used as temporary objects from which to create complex closed objects.
- *Two-dimensional (2D)* or *closedobjects*are objects with boundaries that enclose a region. All closed objects are automatically saved as part of the geometric model when you click **File** > **Save**.
- *Simple* closed objects include thick lines and arcs, circles, and rectangles. A *complex* closed object is one created by joining open objects to enclose an area. For example, to turn an open object into a closed object, draw a line that connects the end points of the open object.

Drawing a Circle in the Layout Editor

Draw a circle by selecting a center point and a radius. Circles are drawn as true surfaces.

1. From the **Draw** menu, select **Primitive** > **Circle**.
2. Select the center of the circle in one of the following ways:
   - Click the mouse on the point.
   - Enter the point's coordinates in the **X** and **Y** fields using the keyboard.
3. Specify the circle's radius in one of the following ways:
   - Select a point on the circle's circumference using the mouse or the keyboard.
   - Type a value for the radius in the **Distance** field and press **Enter**.

The circle is added to the active layer.

Drawing a Rectangle in the Layout Editor

Draw a rectangle by selecting either two diagonally opposite corners.

1. From the **Draw** menu, select **Primitive** > **Rectangle**.
2. Select the first diagonal corner in one of the following ways:
   - Click the mouse on the point.
   - Type the point's coordinates in the **X** and **Y** fields.

To delete the selected point and start over, press **Esc** or right-click and select **Cancel**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

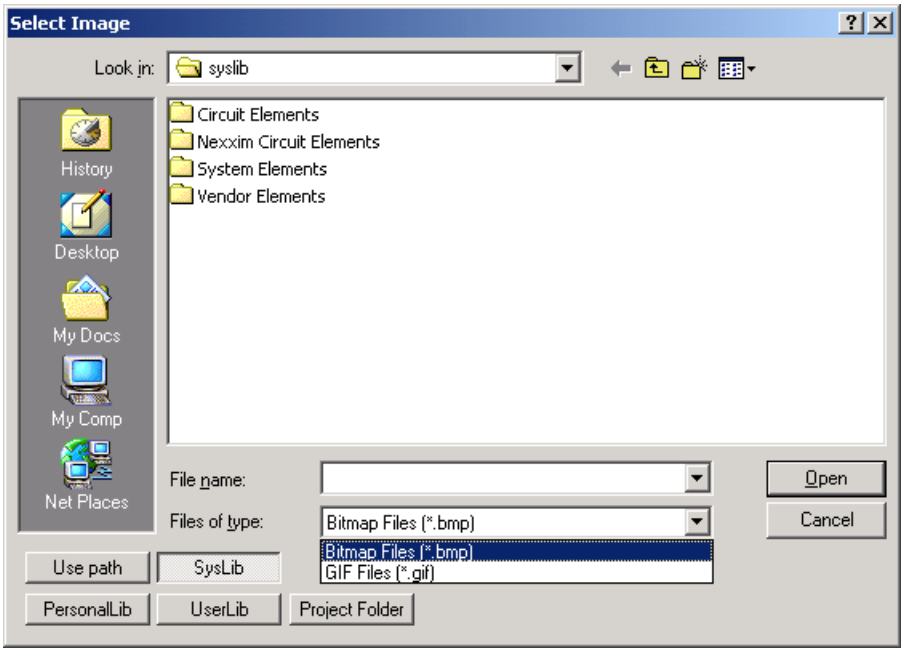
3. Select the second diagonal using the mouse or the keyboard.

The rectangle is added to the active layer.

Drawing an Image in the Layout Editor

Draw an image by using the **Draw** > **Primitive** > **Image** command.

1. From the **Draw** menu, select **Primitive** > **Image** to open the **Select Image** window.



2. Select an appropriate path, Lib, or Folder to access by clicking one of the following buttons: **Use path**, **SysLib**, **PersonalLib**, **UserLib**, **Project Folder**.

- Within the **PersonalLib**, **UserLib**, or **SysLib** folder, select the file from those listed, or type the name of the file in the **File Name** area. Note that **SysLib** is reserved for libraries supplied with HFSS.
- If you select **Use path**, type the name of the file in the **File Name** area, or click the **Look in** field to navigate to the file and record its name. Note that objects imported with **Use Path** may not be portable if the design is moved to a different machine.
- When **In project folder** is selected, references to the file in the design are relative to the directory where the project resides. In this case, the path is saved in the project **.adsn** file as a variable such as:

  .lib '$PROJECTDIR/x_113854.lib'

The variable $PROJECTDIR are expanded to the current location of the project when the design is converted to a netlist and run by an analysis tool. Otherwise, an absolute path is

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

saved. If you move a project and its library files together to a new directory, you can preserve the file references by selecting the **Project Folder** option.

3. Select the file type from one of the following:
- Bitmap Files (*.bmp)
- GIF Files (*.gif)
4. Select a file on the display, or type the name of the file in the **File Name** field.
5. Click **Open** and a ghost image appears that drag over the schematic display and click to place. The image is added to the active layer.

> **Note:**
>
> - The **PersonalLib** folder is located at **<Project Directory>/PersonalLib**
> - The **userlib** folder is located at **<Installation Directory>/userlib**
> - The **syslib** folder is located at **<Installation Directory>/syslib**
>
> **<Project Directory>** is the location you specified for the projects,and **<Installation Directory>** is where HFSS was installed during setup.

Drawing an Arc in the Layout Editor

Draw an arc by selecting a center point, start point, and endpoint. Arcs are drawn counter-clockwise and as true surfaces.

1. From the **Draw** menu, select **Primitive** > **Arc**.
2. Select the center point of the arc in one of the following ways:
- Click the mouse on the point.
- Type the point's coordinates in the **X** and **Y** fields.
3. Select the arc's start point using the mouse or the keyboard.
4. Select the arc's endpoint using the mouse or the keyboard.

The arc is added to the active layer.

> **Note:**
>
> Arcs are drawn as true circles in the Layout Editor, but they are segmented for mesh generation.

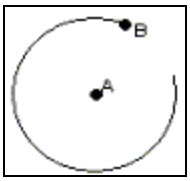
Additional Notes for Drawing an Arc

1. After clicking the arc toolbar icon or the arc menu item, the cursor changes to indicate arc creation. The first mouse-click location in the editor specifies the center point for the arc.

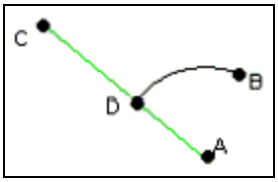
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, rather than clicking in the editor, you can always enter an equivalent value in the coordinate bar to indicate a location.

2. Cursor movements draw an arc showing the radius and starting angle based on cursor position. Both the radius and starting angle may change as the cursor is moved.
3. The next mouse-click in the editor specifies both the radius and the starting angle for the arc. From the following figure, point A is the location of the first click. Point B is the location of the second click.



4. Cursor movements change the ending angle only; the radius is fixed. During a mouse hover, the cursor is moved to the radius. The end angle indicated by the cursor position remains unchanged. Grid snapping is relative to the moved position and the snap is constrained to remain at the defined radius. Only end angles at 0, 90, 180, and 270 degrees on the start angle grid snaps; other end angles do not intersect a grid intersection. From the following figure, the cursor hovers at C. The line from A to C indicates the end angle position. D is the location to which the cursor is moved.



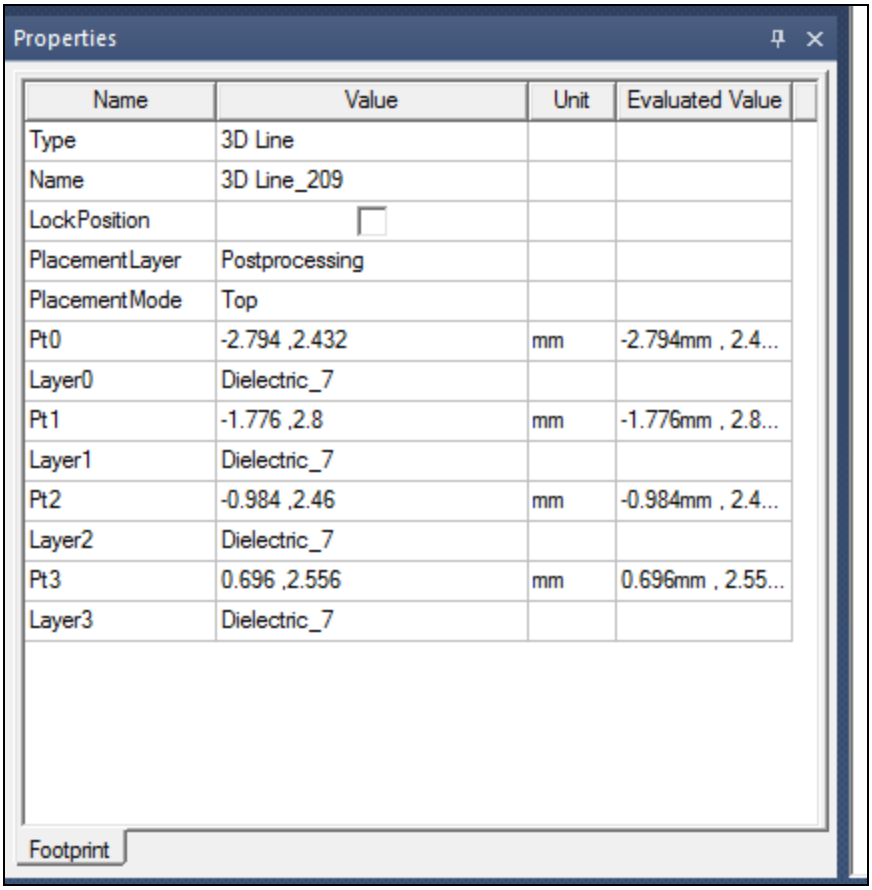
5. The next click specifies the end angle for the arc. Note that this location may NOT be the end point of the arc. The end point of the arc is the position along the ending angle at the defined radius. If the cursor is clicked at C, the arc is created with D as the end point. If the cursor is clicked at D, the arc is created with D as the end point.

Drawing a 3D Line Segment

The following describes how to draw an object with one or more straight segments.

1. **Create the 3D Line**: To draw the line, on the **Draw** menu, select **Primitive** > **3D Line**. Once you have completed the line, its associated property window should appear similar to the following example.

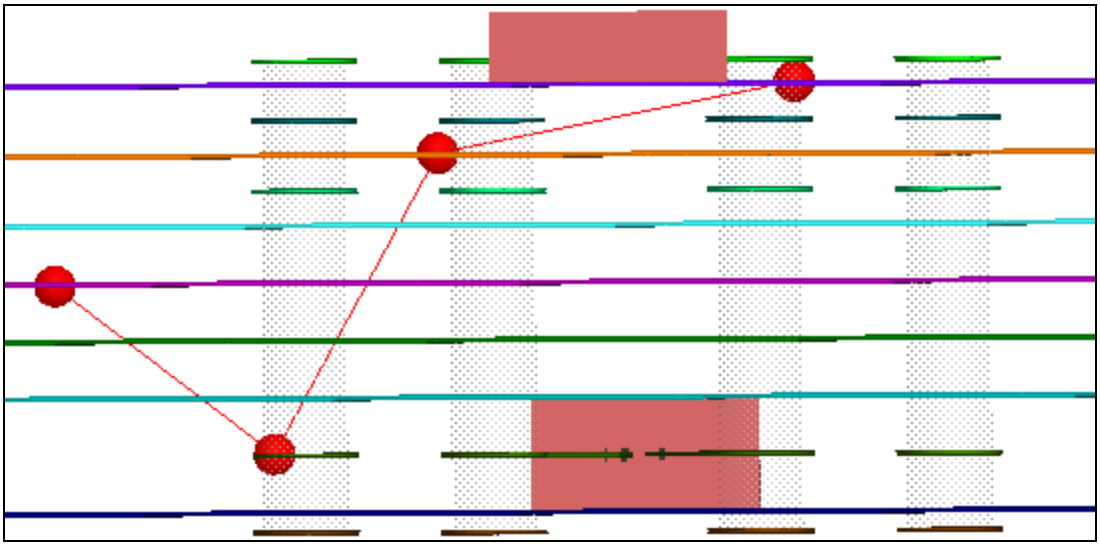
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The property window for the line has following features: — **Placement Layer**: The layer on which this primitive exists. It is always set to 'Post-processing' and cannot be changed. — Each point has its 2D coordinates and a layer on which it is placed currently. — The layer for each point is initially set to the active Layer in the 'Active Layer' drop-down menu. — **PlacementMode**: Can be set to Top, Bottom, Middle, or 3D. It is initially set to Top.

2. **Select the Layers for Each Point**: The field named layer contains a list of all the stackup layers present in the design. The point can be placed on any of those lines. Each layer has a different elevation and changing the layer for each point has the effect 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.

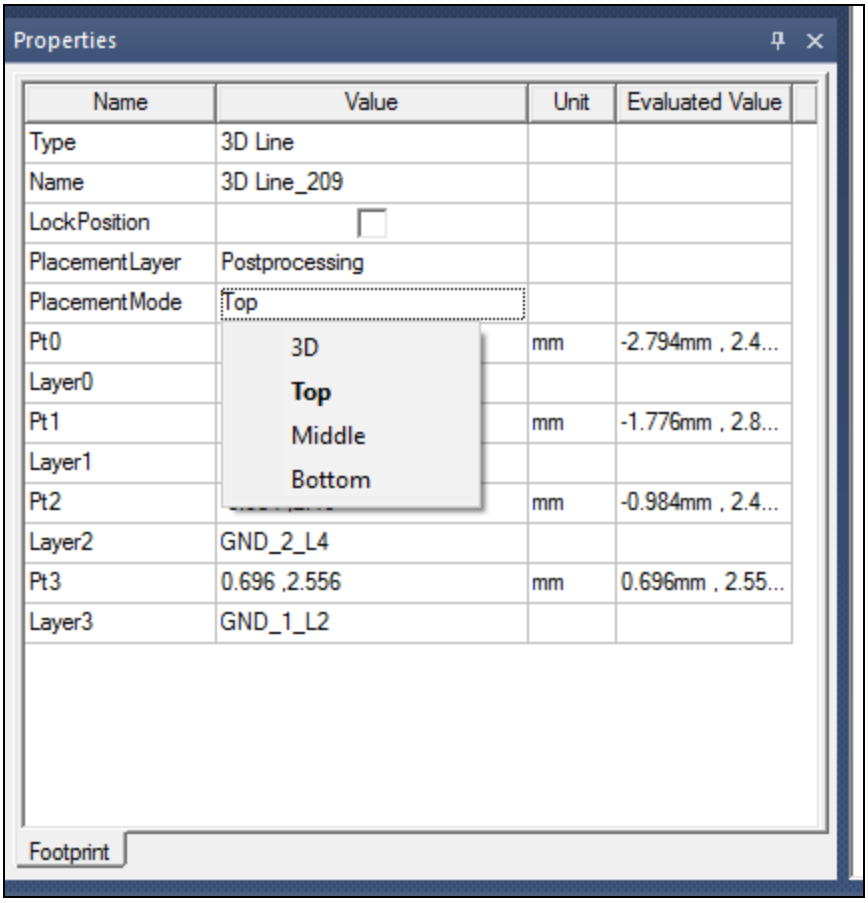


3. **Toggle between placement modes:**

- **Top**: Places the point on the ‘Upper elevation’ of the current layer.
- **Bottom**: Places the point on the ‘Lower elevation’ of the current layer.
- **Middle**: Places the point on the middle elevation (lower elevation + thickness/2) of the current layer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

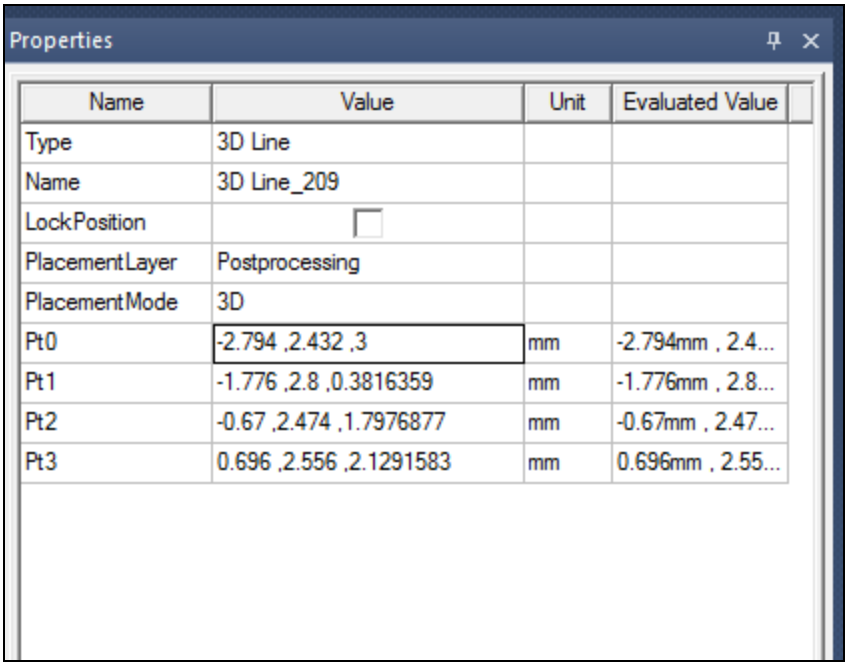
- **3D**: The point has true 3D coordinates which can be changed as selected.



- **Changing from Top to 3D:** The property window shows the layer names have been removed and the ‘Pt’ field contains the true 3D coordinates of the point. The 3D coordinates of the point are currently set to the upper elevation of the layer, since you have toggled from Top to 3D.
- You can change the coordinates of the point using the property window, as shown in the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following image. Note the difference in the Z-coordinate for Pt0.



- In 3D mode, while toggling back to top/bottom/middle, the layer for each point is set to the layer whose elevation is closest to the point. The point is then moved to that layer.
- Toggling from 3D to Middle/Bottom or the reverse yields similar results and assigns layers to the points based on the Middle/Lower elevation.

4. **Script Recording and Replay**:

- 3D Line script record/replay works just like any other script record/replay. Before drawing the line, click on **Tools > Record Script to Project**. Once you have finished drawing the line, click on **Tools > Stop Recording**.
- From the project manager window, go to **Definition >Scripts**. Right-click the script to see the available operations.
- The recorded script looks like the following diagram. Running the script returns the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

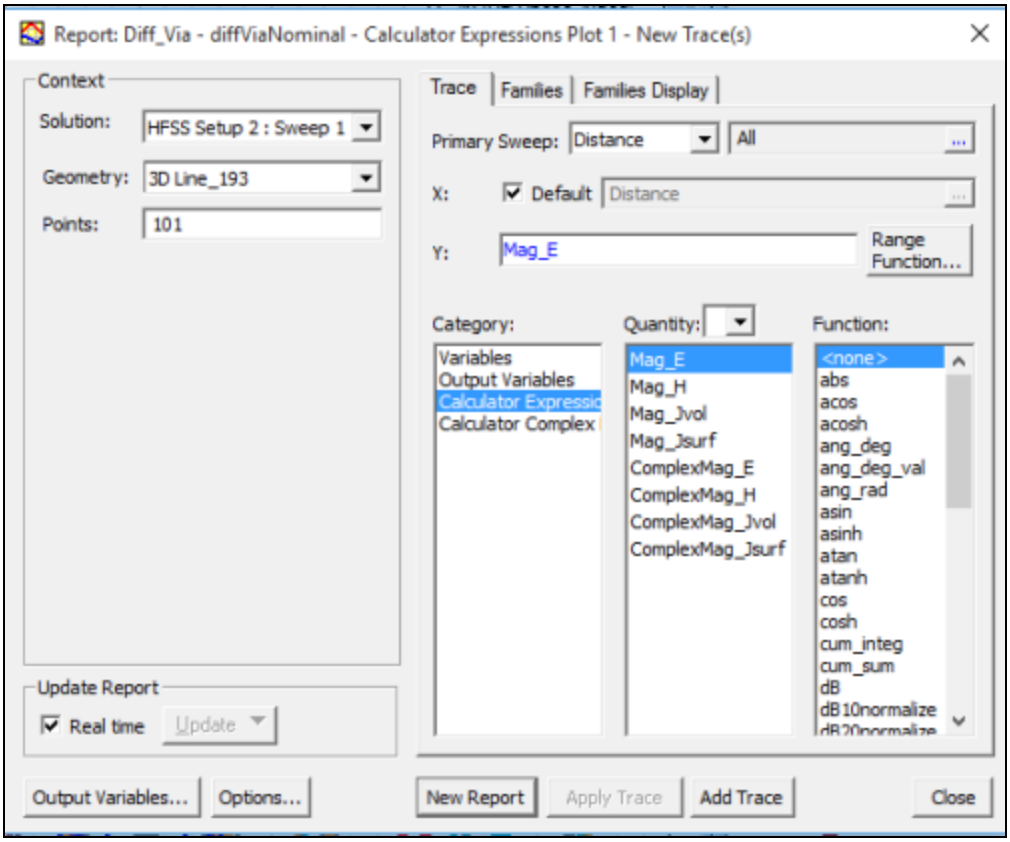
geometry of the line captured in the script.

```
' ----------------------------------------------
' Script Recorded by ANSYS Electronics Desktop Version 2018.1.0
' 10:08:13  Feb 14, 2018
' ----------------------------------------------
Dim oAnsoftApp
Dim oDesktop
Dim oProject
Dim oDesign
Dim oEditor
Dim oModule
Set oAnsoftApp = CreateObject("Ansoft.ElectronicsDesktop")
Set oDesktop = oAnsoftApp.GetAppDesktop()
oDesktop.RestoreWindow
Set oProject = oDesktop.SetActiveProject("Diff_Via")
Set oDesign = oProject.SetActiveDesign("diffViaNominal")
Set oEditor = oDesign.SetActiveEditor("Layout")
oEditor.AddObject "3D Line", Array("NAME:Contents", "3D LineGeometry:=", Array("Name:=", _
  "3D Line_210", "LayerName:=", "Postprocessing", "lw:=", "0", "gm:=", Array("m:=", _
  1, "pnt:=", Array("lid:=", 13, "pos:=", Array("X:=", "-2.69mm", "Y:=", "2.546mm", "Z:=", _
  "0")), "pnt:=", Array("lid:=", 13, "pos:=", Array("X:=", "-1.748mm", "Y:=", _
  "2.792mm", "Z:=", "0")), "pnt:=", Array("lid:=", 13, "pos:=", Array("X:=", _
  "-1.276mm", "Y:=", "2.508mm", "Z:=", "0")), "pnt:=", Array("lid:=", 13, "pos:=", Array("X:=", _
  "0.668mm", "Y:=", "2.688mm", "Z:=", "0")))))
oEditor.ChangeProperty Array("NAME:AllTabs", Array("NAME:BaseElementTab", Array("NAME:PropServers", _
  "3D Line_210"), Array("NAME:ChangedProps", Array("NAME:Layer1", "Value:=", _
  "Inner_Layer_3_L10"))))
oEditor.ChangeProperty Array("NAME:AllTabs", Array("NAME:BaseElementTab", Array("NAME:PropServers", _
  "3D Line_210"), Array("NAME:ChangedProps", Array("NAME:Layer2", "Value:=", "GND_2_L4"))))
oEditor.ChangeProperty Array("NAME:AllTabs", Array("NAME:BaseElementTab", Array("NAME:PropServers", _
  "3D Line_210"), Array("NAME:ChangedProps", Array("NAME:Layer3", "Value:=", "GND_1_L2"))))
oEditor.ChangeProperty Array("NAME:AllTabs", Array("NAME:BaseElementTab", Array("NAME:PropServers", _
  "3D Line_210"), Array("NAME:ChangedProps", Array("NAME:PlacementMode", "Value:=", "3D"))))
oEditor.ChangeProperty Array("NAME:AllTabs", Array("NAME:BaseElementTab", Array("NAME:PropServers", _
  "3D Line_210"), Array("NAME:ChangedProps", Array("NAME:Pt0", "X:=", "-2.69mm", "Y:=", _
  "2.546mm", "Z:=", "3mm"))))
oEditor.ChangeProperty Array("NAME:AllTabs", Array("NAME:BaseElementTab", Array("NAME:PropServers", _
  "3D Line_210"), Array("NAME:ChangedProps", Array("NAME:PlacementMode", "Value:=", "Top"))))
```

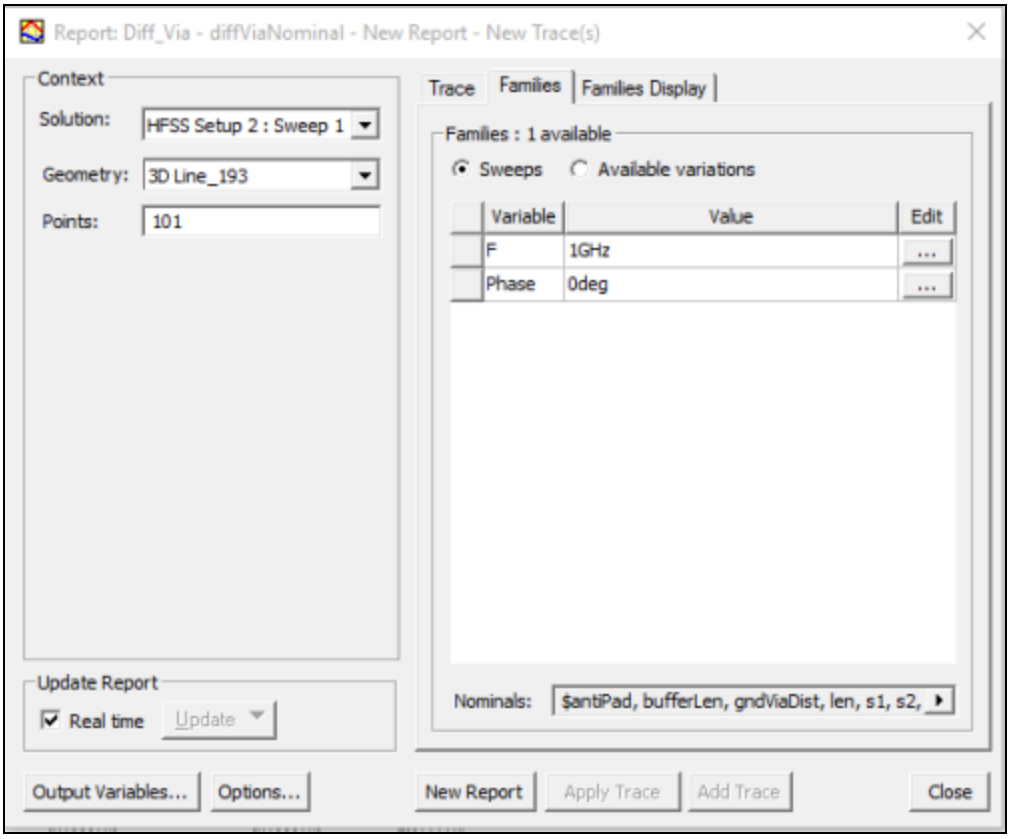
5. **Analyzing the Results and Plots**:

- Create an HFSS Setup, then analyze.
- Draw a 3D line by clicking on the icon in the ribbon.
- From the property window, change the placement mode to 3D. From the toolbar, change the moving axis to Z. Then change the coordinates of some points in the line.
- Select **Results** > **Create Fields Report** > **Rectangular plot** and ensure all the settings are the same as in the following diagram that shows the settings in the **Trace** and **Families** tabs.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

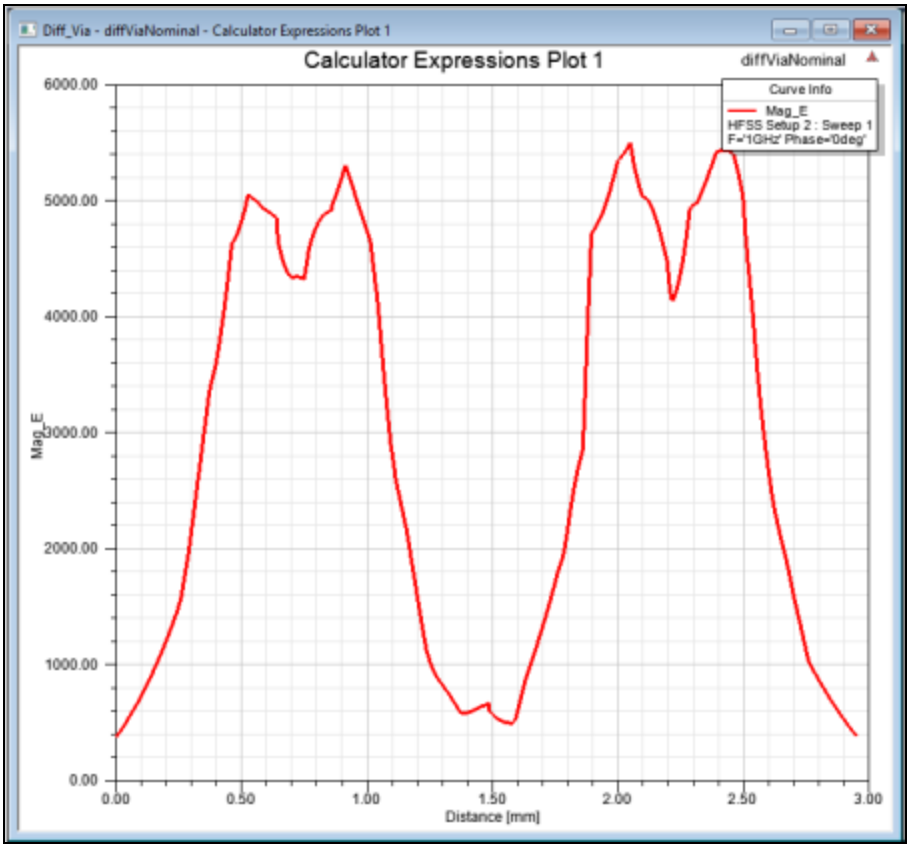


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The resulting plot is 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.



Now return to the **Layout Editor** and move the line to a new position. Observe the plot should change accordingly.

In a similar manner to these examples shown, other types of plots can be obtained by right-clicking on **Results** in the **Project Manager** window and choosing on the **Create Standard Report** drop-down menu.

For more information, refer to the plotting and report information listed in Post Processing and Generating Reports and Post Processing and Generating Reports for 2D and Circuit.

Drawing a Line in the Layout Editor

Draw an object with one or more straight segments using the **Draw** > **Primitive** > **Line** command.

1. From the **Draw** menu, select **Primitive** > **Line**.
2. Select the first point of the line in one of the following ways:
   - Click the mouse on the point.
   - Type the point's coordinates in the **X** and **Y** fields.
3. Select the next point of the line using the mouse or the keyboard. Notice that HFSS draws a line between the two points.

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, press **Backspace** or right-click and select **Back Up**.
- To delete all selected points and start over, press **Esc** or right-click and select **Cancel**.

4. Repeat step 3 to enter each point.
5. Complete the line in one of the following ways:
   - Double-click the final point of the line.
   - Press **Enter** or right-click and select **Finish**.

The line is added to the active layer.

You can edit the line's properties in the **Property Control** window to change the line's width, bend type, and end type in the **Line Styles** window.

> **Note:**
>
> HFSS no longer displays the "Linewidth" property for primtives in the Properties Window. In order to make modifications to the linewidth, you must use the "Expand" function. Note that a zero-width line is ignored by the Meshmaker.

Stitch Lines

Click the **Layout** > **Stitch Lines** command to stitch together connected/crossing lines into polygons and lines. Selected lines with zero width are united to form polygons when the segments close. Selected zero-width lines are first split at all points of intersection. Then lines are stitched together into polygons. Stitching is done in such a way as to avoid forming lines that cross.

Specifying Line Width in the Layout Editor

HFSS no longer displays the **Linewidth** property for all primitive types in the **Properties** window. For trace primitives, the line width can still be adjusted directly through **Properties**, but for other primitive types, such as polygons, use the **Expand** tool to modify the width instead.

> **Note:**
>
> A zero-width line is ignored by the Meshmaker.

Legacy adsn Files

The following apply to opening legacy adsn files or importing files of other formats that contain polygons, circles, and rectangles with linewith specifications:

- Polygons with linewidth are converted to non-linewidth polygons of the same shape.
- Rectangles with linewidth are converted to non-linewidth polygons of the same shape.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Circles with linewidth are converted to non-linewdith circles. Circle diameters are equal to the diameters of the legacy circles plus the linewidth.

Legacy Script Commands

The following apply to executing legacy script commands that operate on polygons, circles, and rectangles with linewidths:

- All linewidth properties of rectangles is ignored and warnings are posted in the Message Manager window.
- All polygons and circles are converted to corresponding non-linewidth polygons and circles, but the same shapes are generated.

Reverse Line Command in the Layout Editor

The Reverse Line command reverses the direction of selected Bond Wires, Lines and Arcs. The command is accessible on the Draw menu, the Toolbar, and the right-click Draw drop-down menu when an object is selected. Invalid objects is ignored.

Drawing a Polygon in the Layout Editor

A polygon is an arbitrarily shaped, closed object with three or more sides.

1. From the **Draw** menu, select **Primitive** > **Polygon**.
2. Select the first point of the polygon in one of the following ways:
   - Click the mouse on the point.
   - Type the point's coordinates in the **X** and **Y** fields.
3. Select the next point of the polygon using the mouse or the keyboard. Notice that HFSS draws a line between the two points.
   - To delete the last point that was entered, press **Backspace** or right-click and select **Back Up**.
   - To delete all selected points and start over, press **Esc** or right-click and select **Cancel**.
4. Repeat step 3 for each point. HFSS will draw a line between the first point and last point entered to close the object.
5. Complete the polygon in one of the following ways:
   - Double-click the final point of the line.
   - Press **Enter** or right-click and select **Finish**.

The polygon is added to the active layer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

HFSS no longer displays the "Linewidth" property for primitives in the **Properties** window. In order to make modifications to the linewidth, you must use the "Expand" function. Note that a zero-width line is ignored by the Meshmaker.

Curving Polygon Edges in the Layout Editor

1. Follow steps **1-5** in Drawing a Polygon in the Layout Editor.
2. Hold **Ctrl** and drag the center handle of an appropriate edge to curve it.

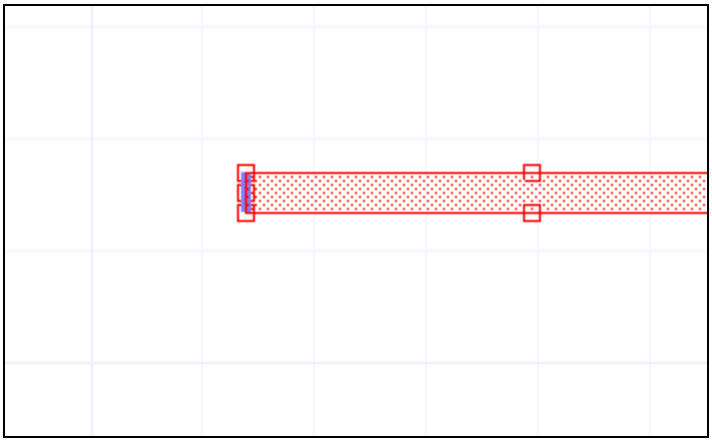
Drawing a Void in the Layout Editor

Follow these steps to create a void, or remove material from an object in the shape of a circle, rectangle, polygon, or line.

Note:

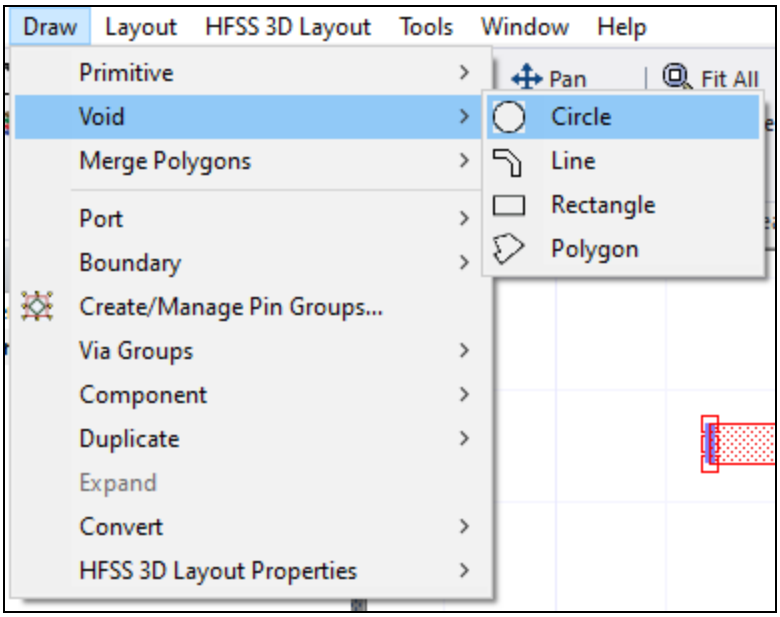
One object can have multiple voids but each void can only be associated with one parent object (e.g., to create two concentric rings, draw the outer circle, then add a circular void within it).

1. Select a parent object.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Navigate to **Draw**, then select **Void** and one of the following:

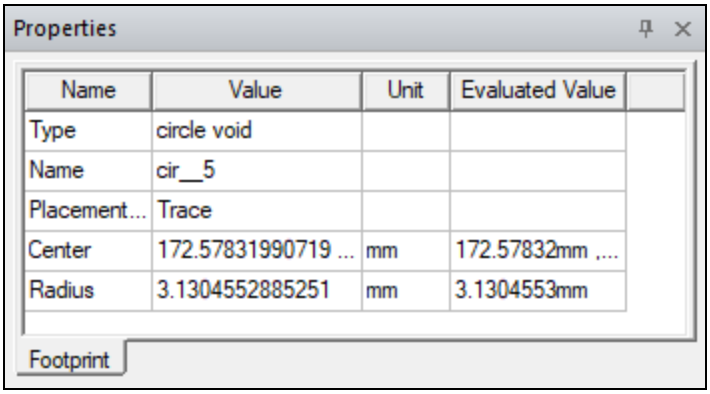


- **Circle**. Then follow the procedure for drawing a circle.
- **Rectangle**. Then follow the procedure for drawing a rectangle.
- **Polygon**. Then follow the procedure for drawing a polygon.

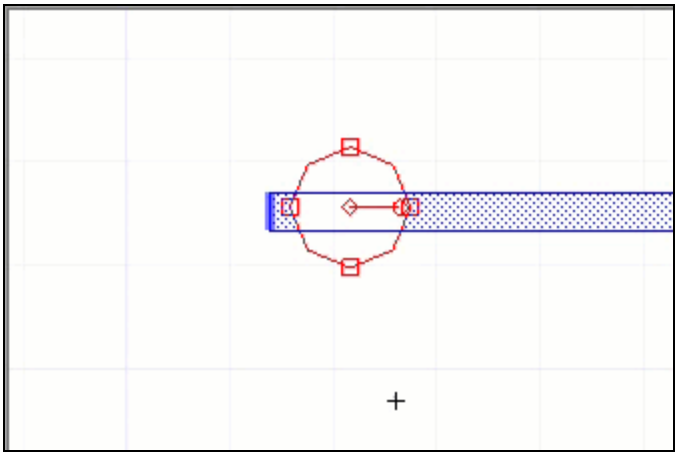
> **Note:**
>
> To **Remove a Line**, navigate to **Draw** menu, then select **Void** > **Line**. Follow the procedure for drawing a polygon. A void cannot be created in the shape of a line on an existing line object. Material is removed from the object in the shape and dimensions of the void.

2. Repeat steps **1-2** to create additional voids in an object.
3. View or edit a void's properties in the **Property Control** window or use the mouse to drag the void's handles.

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+drag** the void's handles to change its shape.



Note:

3D vias may contain voids. Voids in 3D vias can be placed entirely or partially in the outer contour. No attempt is made to geometrically align voids in the upper polygons with those in the lower. The order in which voids on different layers are created is used to control their alignment.

Drawing a Via in the Layout Editor

A via is an object that extends in the z direction and connects two signal layers, allowing a signal to travel from one layer to another. Unlike cross-layer plates, vias have volume, can be electrically large, and vary in shape. For these reasons, vias also use greater computational resources than cross-layer plates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vias are created from two 2D objects. Each 2D object is located on a trace layer. The via is extruded between the 2D objects when a mesh is generated, thereby connecting the layers.

> **Note:**
>
> Vias can also be 2.5D, 3D, or created from primitives drawn on a padstack layer (i.e., circles, polygons, arc, or lines).

2.5D Vias

2.5D vias must have a circular cross-section and be strictly vertical. A 2.5D via has a finite radius and the current is assumed to be flowing along the surface of the via. The current flow is assumed strictly vertical; the Ansys Electronics Desktop (AEDT)™ electronics systems design platform treats the via as a 1D wire connecting the two layers. All 2.5D vias have a default complex load impedance of zero.

3D Vias

3D vias can be any shape, but the 2D objects they connect must have the same number of edges. The current is assumed flowing on the surface of the via in all directions.

> **Note:**
>
> 3D vias may contain voids. Voids in 3D vias can be placed entirely or partially in the outer contour. No attempt is made to geometrically align voids in upper polygons with those in lower polygons. Instead, the order in which voids on different layers are created is used to control their alignment.

Drawing a Cross-Layer Plate in the Layout Editor

After selecting two or more objects on different layers, navigate to **Draw** and select **Create 3D Structure** to connect the objects and create a single 2D or 3D object. The selected objects must possess the same number of points. If the selected objects contain open lines, the created structure will include a two-dimensional face (i.e., cross-layer plate) that spans the different layers. If the selected objects contain closed lines, the structure will include a three-dimensional solid (i.e., 3D Via) that spans the different layers. These 3D surfaces are included in the model. 3D structures also possess material attributes.

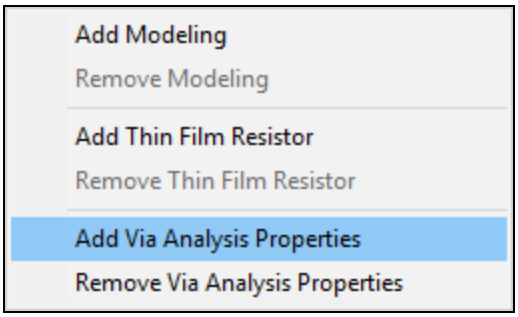
Drawing a Hole in the Layout Editor

A hole is a placeholder created while drawing a via. To create a void, or remove material from an object, refer to Drawing a Void in the Layout Editor.

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 2.5D Via in the Layout Editor

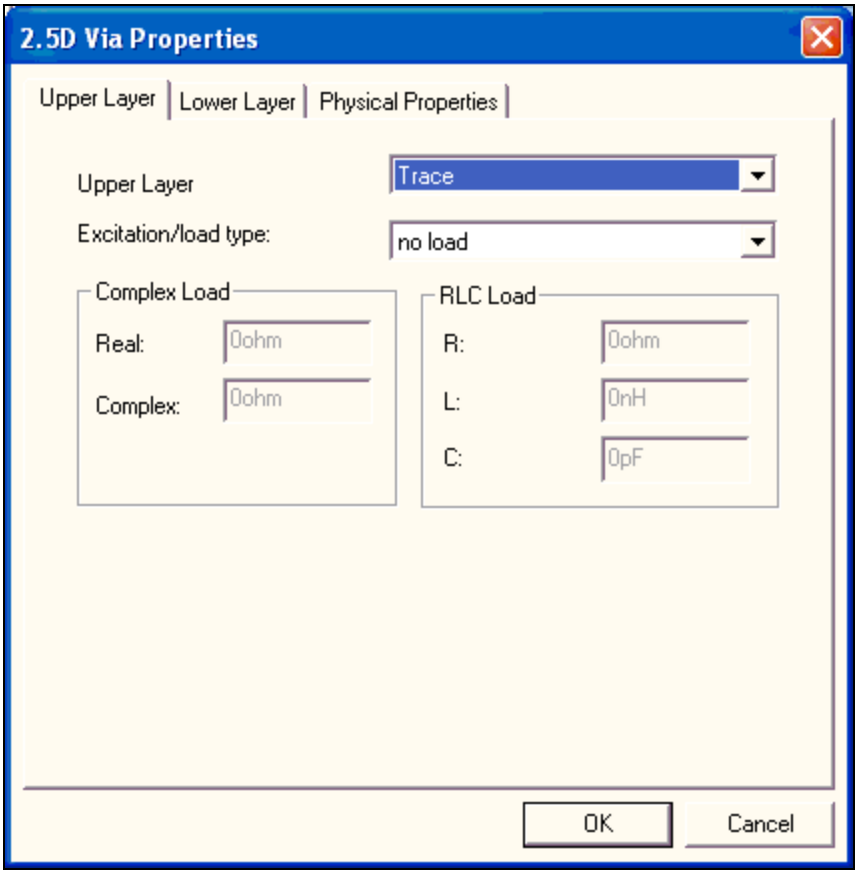
1. Select a signal or ground layer to draw upon.
2. From the **Draw** menu, select **Via**.
3. Select the center point of the via using the mouse or the keyboard.
4. With a via selected, click **Draw** > **HFSS 3D Layout Properties**, or right-click the via and select **HFSS 3D Layout Properties**, which brings up the following drop-down menu:



5. Select **Add Via Analysis Properties** to open the **EM Design** tab of 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.

window.



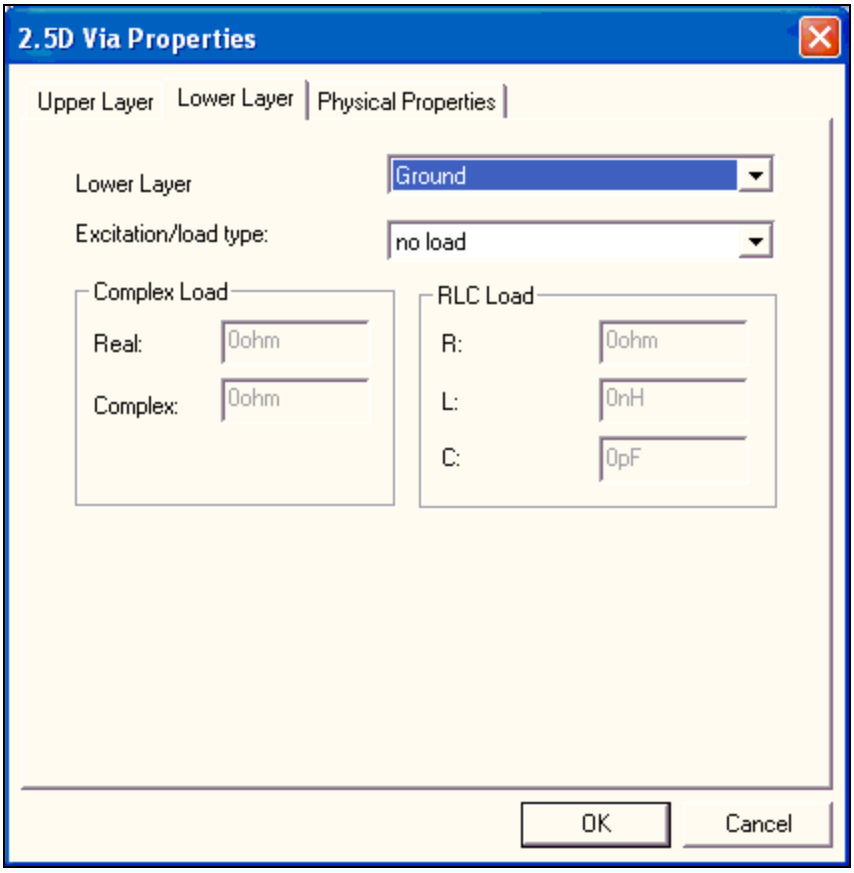
Upper Layer Properties

6. Click the **Upper Layer** tab and do the following:
    a. Specify the top layer — a signal or ground layer — at which the via terminates.
    b. If the top layer contains the load, select a load type on the **Excitation/Load Type** area. You may only define a load at one end of the via. The load on the other end is then set to zero.
    c. If you selected **complex**, type the real part of the complex load in ohms in the **Real** field. Then type the imaginary part of the complex load in ohms in the **Complex** field.
    d. If you selected any RLC combination, do the following:
        - Type the resistance value in ohms in the **R** field. It must be a positive or zero value.
        - Type the inductance value in nanohenries in the **L** field. It must be a positive or zero value.
        - Type the capacitance in picofarads in the **C** field. It must be a positive value.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lower Layer Properties

7. Click the **Lower Layer** tab and do the following:

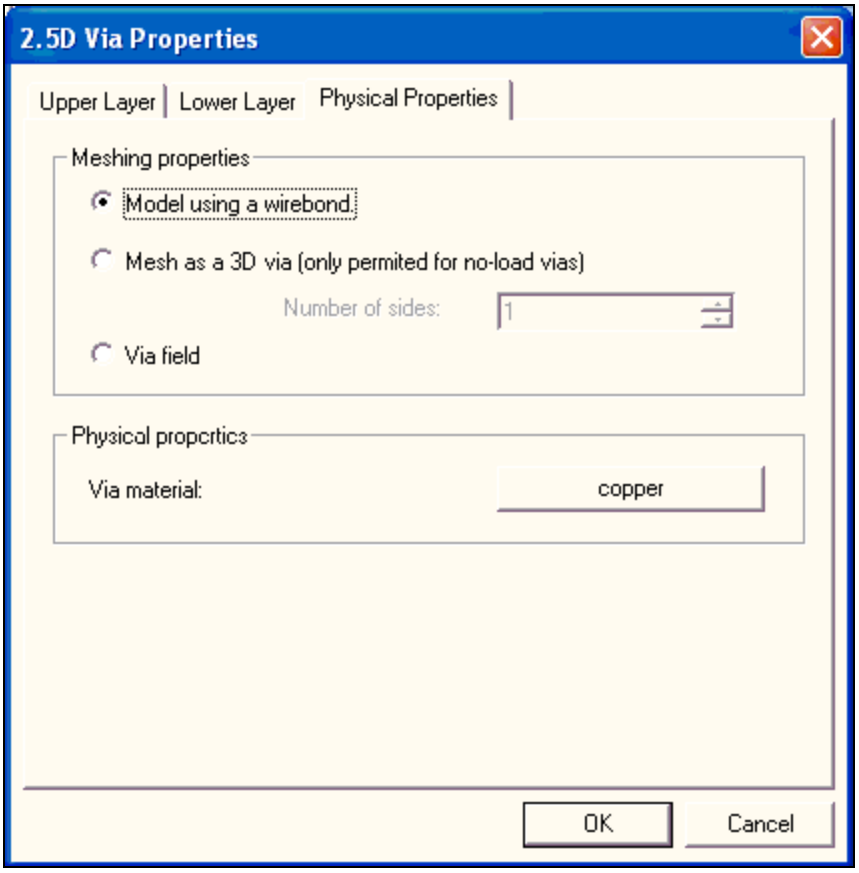


   a. Specify the bottom layer — a signal or ground layer — at which the via terminates
   b. If the bottom layer contains the load, select a load type on the **Excitation/Load**
   c. If you selected **complex**, type the real part of the complex load in ohms in the **Real** field. Then type the imaginary part of the complex load in ohms in the **Complex** field.
   d. If you selected any RLC combination, do the following:
      - Type the resistance value in ohms in the **R** field. It must be a positive or zero value.
      - Type the inductance value in nanohenries in the **L** field. It must be a positive or zero value.
      - Type the capacitance in picofarads in the **C** field. It must be a positive value.

Physical Layer 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 the **Physical Properties** tab and do the following:



   a. Optionally, select **Model using a wirebond**
   b. Optionally, select **Mesh as a 3D Via**, then select the **number of sides**
   c. Optionally, select **Via field**
   d. Click **Via material** to open the material browser window. Follow the procedure for assigning a material.
9. Click **OK**. The via is set at the specified position.

> **Note:**
>
> When working with a Via, select **Draw** > **Toggle Between Via and Pin** to convert the Via to a Pin and then reconfigure the Pin settings using the options described in Creating a Probe Port.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Load Types for 2.5D Vias

When defining the properties of a 2.5D via, specify the type of load being applied to the signal or negative signal layer. This is done in the **Via Properties** window, under the **Upper Layer** or **Lower Layer** tab, depending on which layer contains the load. Only the load at one end of the via can be defined. The load on the other end is then set to zero.

Select one of the following load types on the **Excitation/Load Type** area:

complex	A complex number with a real and imaginary part.
RLC series	The resistor, inductor, and capacitor are in series.
RL series, C parallel	The resistor and inductor are in series, and they are in parallel with the capacitor.
RC series, L parallel	The resistor and capacitor are in series, and they are in parallel with the inductor.
LC parallel, R series	The inductor and capacitor are in parallel, and they are in series with the resistor.
LC series, R parallel	The inductor and capacitor are in series, and they are in parallel with the resistor.
RC parallel, L series	The resistor and capacitor are in parallel, and they are in series with the inductor.
RL parallel, C series	The resistor and inductor are in parallel, and they are in series with the capacitor.
RLC parallel	The resistor, inductor, and capacitor are in parallel.
coaxial excitation	The via is an external coaxial probe source that excites the structure as a port. A probe source represents a coaxial-type connector with its center pin extending from the negative signal plane to any signal layer.

If **complex** is selected, specify the real and imaginary portions of the load in ohms.

If any RLC combination is selected, specify the resistance value in ohms, the inductance value in nanohenries, and the capacitance in picofarads.

If **coaxial excitation** is selected, specify the selected current in amps, the phase in degrees, and the coaxial source impedance in ohms.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Vias, Pins, and Padstack Definitions from Primitives

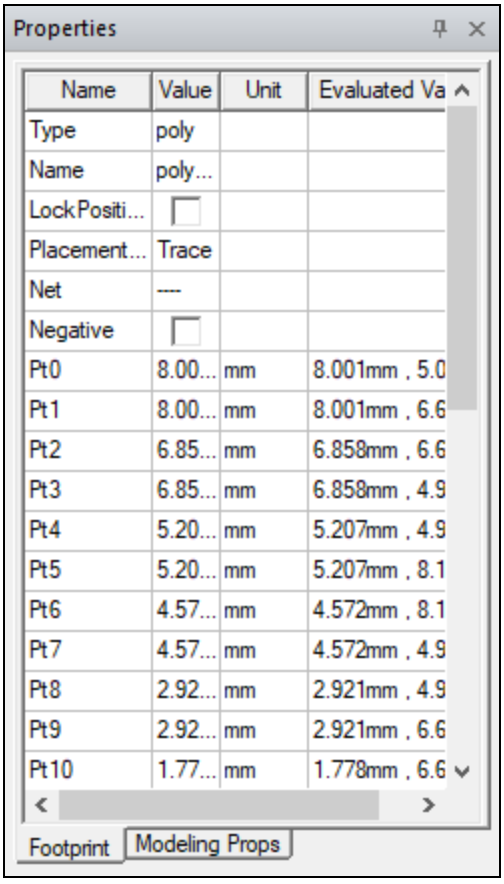
HFSS 3D Layout can create a via or pin from a primitive. When it does, HFSS 3D Layout creates a new padstack definition based on the geometry of the primitive. HFSS 3D Layout then replaces the primitive with a via that is assigned this new padstack definition. To make a primitive into a pin, convert the primitive to a via, the via into a pin.

Multiple primitives can be converted into vias at once. When identical primitives are converted at the same time, only one padstack definition is created. That definition is then applied to all the identical vias. These vias has the same dimensions. If the primitives are not identical, then one padstack definition is created for each unique geometry.

To create vias from primitives, or pins from vias, perform the steps in the following subsections.

To Create Vias from Primitives:

1. Draw one or more primitives. From the **Properties** window, the primitive's **Type** is whatever type of primitive that it is (e.g., "**poly**").

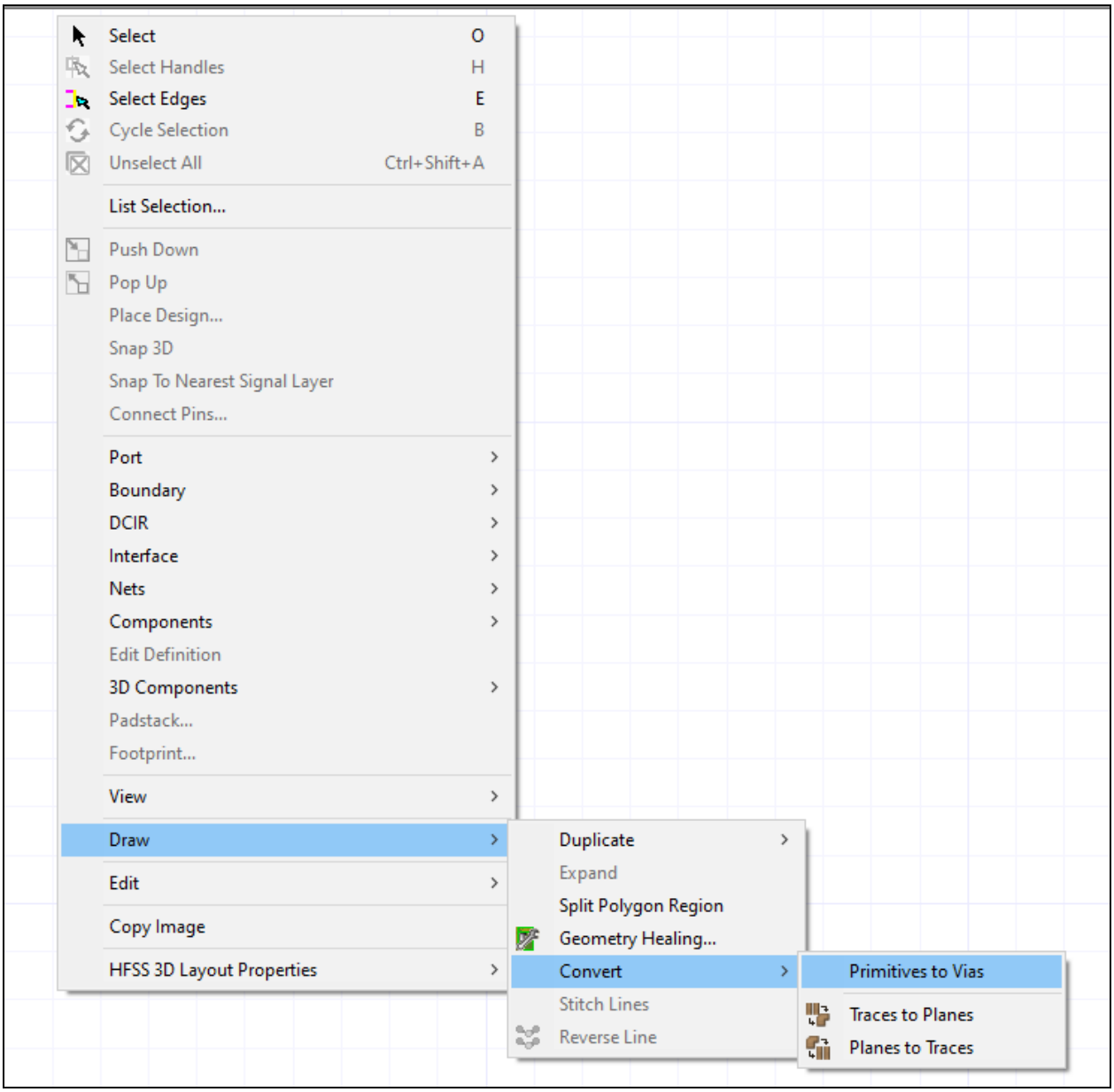


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Highlight the primitives.

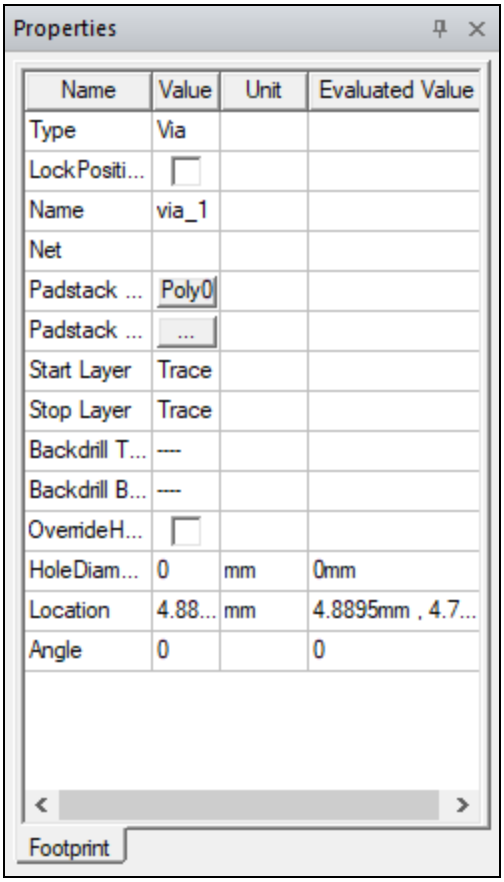
> **Note:**
>
> The vias remain selected until purposefully deselected.

3. From either the menu bar or by right-clicking in the **Layout Editor**, select **Draw** > **Convert** > **Primitives to Vias**.



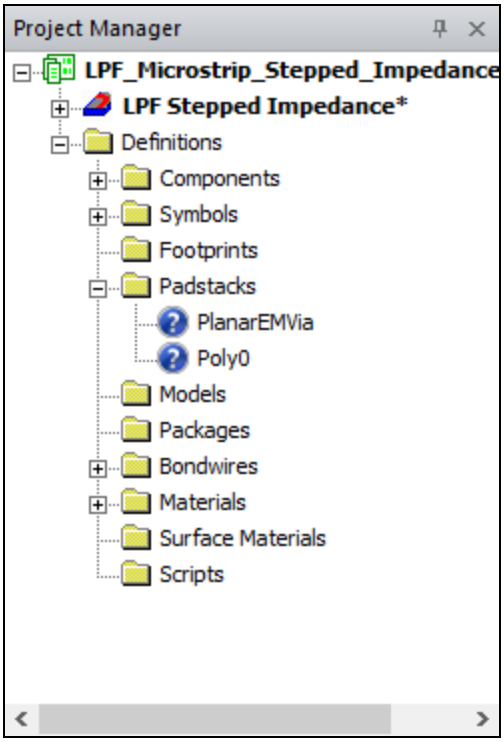
From the **Properties** window, the primitive's **Type** is now **Via**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



New padstack definitions appear in the **Project Tree** (**Definitions** > **Padstacks**) .

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 Pins from Vias:

To convert the new vias to pins, change the **Type** to **Pin** in the **Properties** window.

To See the Geometry of a New Via:

1. From the **Properties** window, click **Padstack Definition** to open the **Select Definition** window. The via's padstack definition is highlighted.The padstack definition is named after the type of primitive.
2. Highlight the new via.
3. Click **Edit Padstack**. The geometry of the via is listed in the **Layers** section.

> **Note:**
>
> If this procedure is attempted on anything that is not a primitive, then it is ignored.

Compressing Primitives in the Layout Editor (IC Mode Only)

Compressing primitives into collections reduces redundancy, optimizes memory, and improves performance by minimizing the number of unique entities that must be stored and

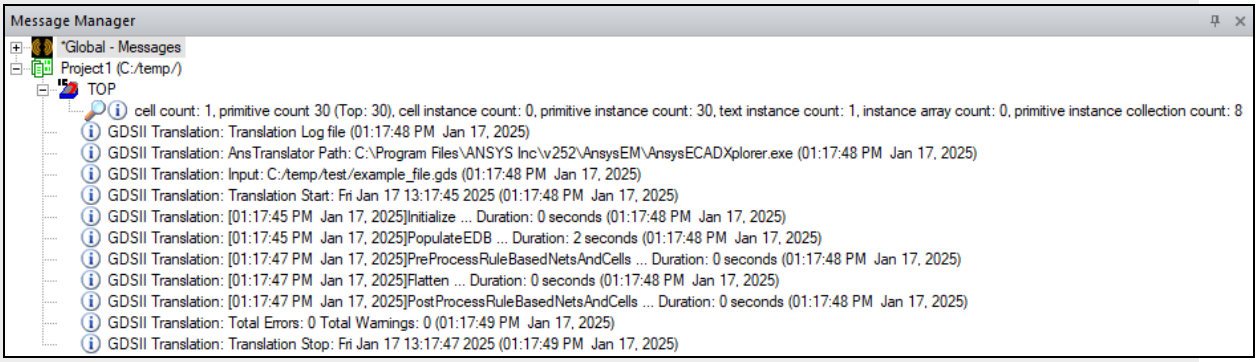
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

processed. Primitives must be geometrically identical rectangles, circles, or convex polygons on the same net, and cannot contain any of the following features:

- Holes or other complex features
- Modeling properties or specific parameters
- Pins or electrical connections
- Membership in any via groups

Note:

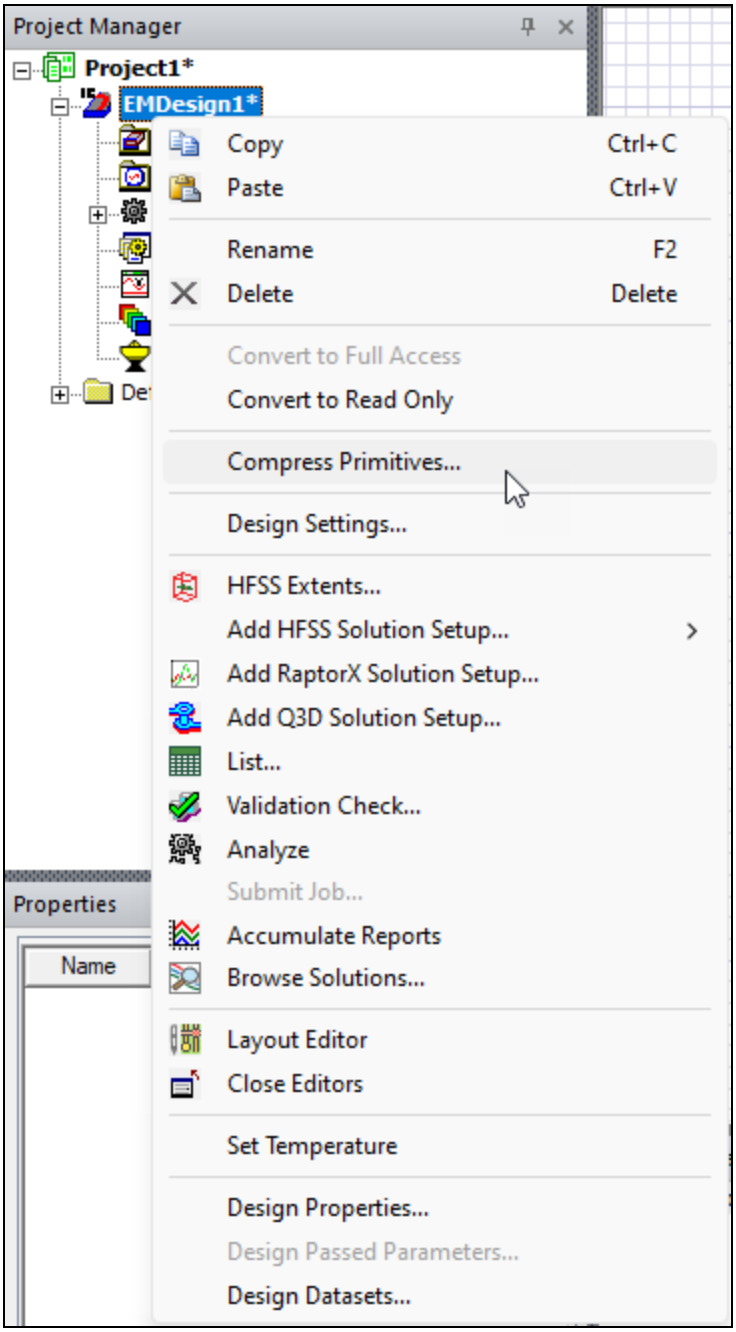
Primitives created by GDSII import are automatically compressed, as evidenced in the **Message Manager** window.



Follow these steps to compress primitives.

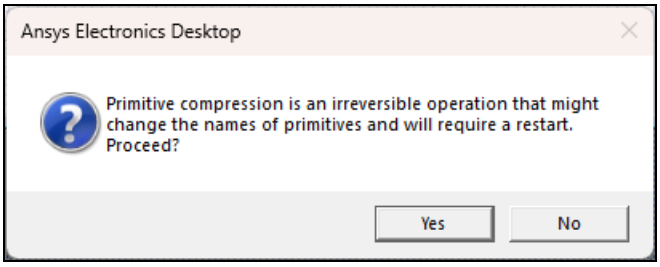
1. From the **Project Manager** window, expand the **Project Tree** (e.g., **Project1**) and right-click the HFSS 3D Layout design (e.g., **EMDesign1**). Then click **Compress Primitives**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

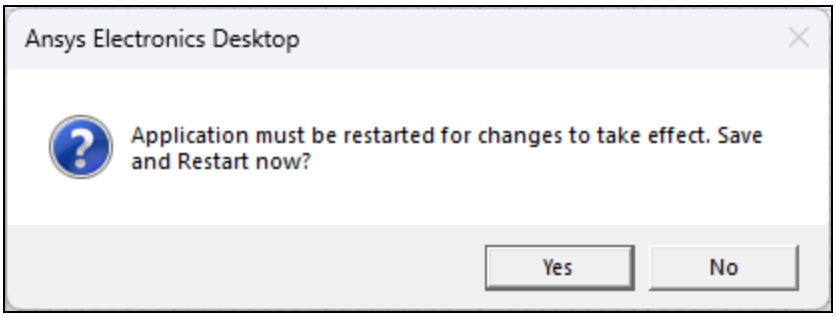


2. Two consecutive prompts will appear. The first warns users that primitive compression is an irreversible operation that may change the names of primitives, and that a restart is required to see an immediate performance improvement. Click **Yes** to compress primitives and continue to the next prompt.

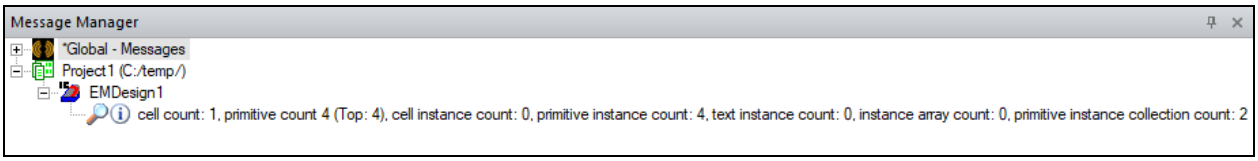
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **Yes** to automatically save changes and restart immediately.



4. After the software restarts, reopen the project file. The **Message Manager** window will display evidence of the compression, including a count of affected elements, including the number of primitive instance collections (e.g., **2**).



Via Clustering (Via Groups)

On-chip inductors are commonly fabricated using thousands of small vias between signal layers. In **Layout Editor**, vias are represented as square primitives. While this is important for the purpose of fabrication, this level of complexity often introduces inappropriate mesh density for simulation. Vias are grouped and represented by a single primitive if the via grouping option is selected. This greatly reduces mesh complexity with a minimal decrease in accuracy. The accuracy is preserved by calculating an equivalent material based on the original geometry.

Given a group of square primitives, the via grouping tool automatically detects clusters of rectangular primitives and creates via groups accordingly. Via groups can then be combined or dissolved as needed.

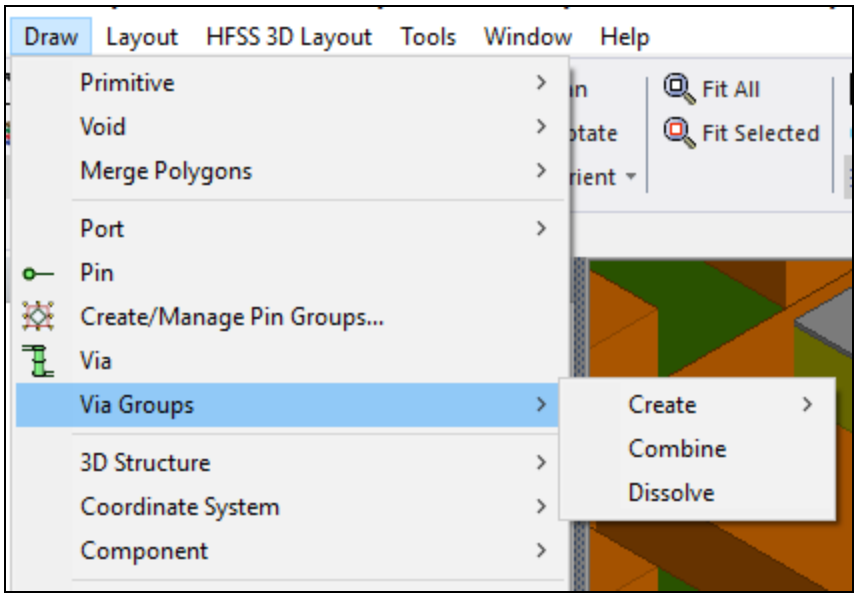
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Both persistent and non-persistent via groups can be created.
- Persistent via groups keep the original primitives of a via group in the layout.
- Non-persistent via group creation deletes the primitive inputs.

Via grouping is accessible through both the user interface and EDB API.

To perform a via grouping:

1. Select the appropriate square primitives.
2. Navigate to **Draw** > **Via Groups** as in the following figure.
3. One of several options can then be selected:
    a. **Create**: Make either persistent or non-persistent via groups on the selected primitives.
    b. **Combine**: Merge selected via groups and/or via primitives into a new via group.
    c. **Dissolve**: Restore the selected via group into the primitives that comprise it.

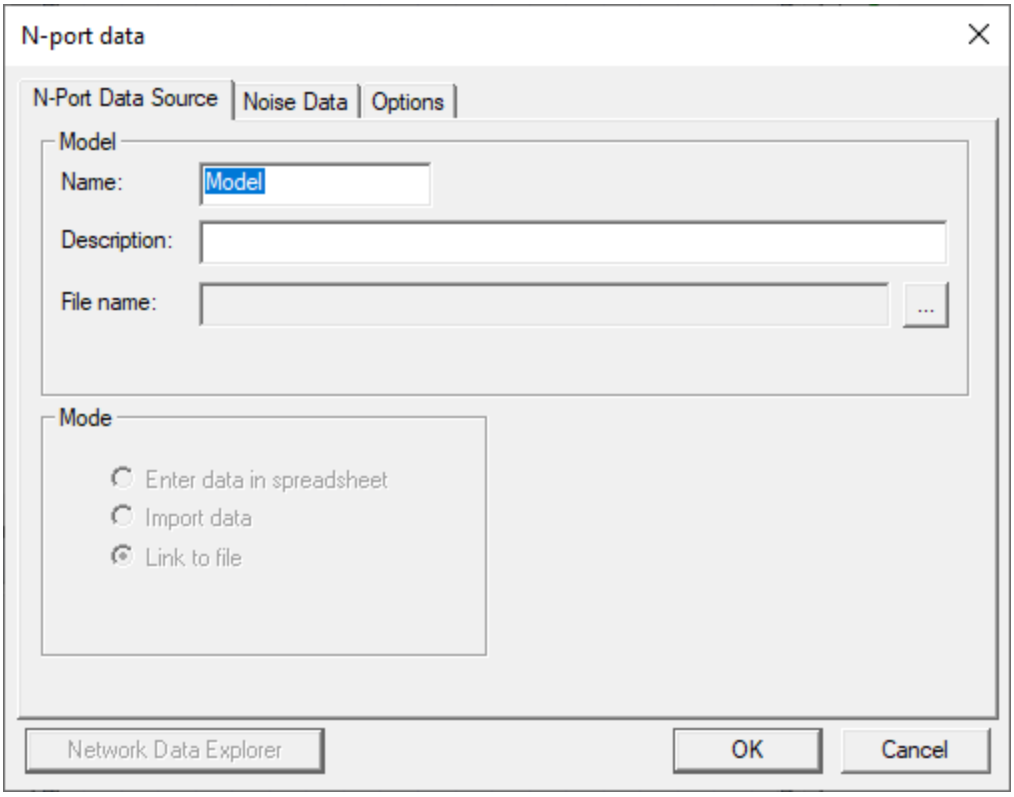


Inserting N-Port Data

To insert N-port data into the active project, follow this general procedure:

1. From the **Project** menu, select **Add Model** > **Add NPort Model** to open the **N-port 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.

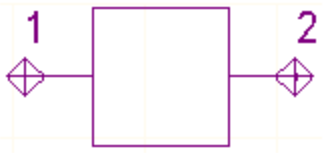


2. Under the **N-Port Data Source** tab, type a name for the N-port in the **Name** field.
3. Select an interpolation method - **Linear** interpolation, interpolation using a **Cubic spline**, or interpolation using a **Rational polynomial** - from the list. Your selection determines how S-parameters between frequency points in your N-port file, if needed, are calculated.
4. Specify the data source:

- Select **Enter data in spreadsheet** if you want to manually enter the S-, Y-, or Z-matrix data for the N-port. Then see Manually Entering N-Port Data.
- Select **Import data** if you want to import the N-port data file into the active project. Then see Importing or Linking to N-Port Data.
- Select **Link to file** if you want to refer to an N-port data file during the solution process. This is useful when you want to avoid importing an N-port data file into the project every time it is updated. Then see Importing or Linking to N-Port Data.

5. Click the **Port Impedance** tab. If selected, change the reference impedance for the S-parameter data.
6. If selected, specify a phase rotation for the S-parameter data.
7. Click **OK** to return to the **Layout Editor**. The N-port is represented by the following symbol, which varies depending upon the number of ports.

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 an arbitrary point in the **Layout** window to place the N-port data in the project.

The N-port is listed under N-Ports in the project tree.

After inserting the N-port into the project, you must draw the connection on the model to the N-port.

Guidelines for Inserting N-Ports

Before inserting an N-port in a project, review the following information:

- The S-parameter file for the N-port must be in one of the following formats:

.szg	Ansys HFSS version 6 and later and Maxwell Strata from version 1.1.
.s *n* p, .tou, .y*p, .z*p	Touchstone .s*n*p, .tou, .y*p, and .z*p format files. For .s*n*p files, the number of ports in the S-parameter file is indicated by *n* (e.g., a Touchstone file with one port has the extension *.s1p). If the S-parameter file has more than 10 ports, the *.tou extension is used.
.flp	HFSS version 2014.
.cit	CITIfile version 1

- N-ports are assumed to be non-radiative. The coupling between the N-ports and the rest of the circuitry is also ignored. Therefore, if you generate an output far-field or near-field file for a structure containing N-ports, the results are contain the radiation on the N-ports.
- Be aware the order in which you connect the model ports to the N-port ports affects the solution. When you select the ports to connect to the N-Port in the **Layout** window, the order in which you select them determines the order of port connection. It is very important to connect the right ports since most active device S-parameters are not symmetrical.
- If your N-port is an active or nonlinear device and you are inserting the device along a section of microstrip line, it is very important the dimensions of the microstrip are as precise (as close to the actual, physical model) as possible. This precision increases the solution's accuracy when connecting the complex S-parameters of the N-port to the overall circuit.
- The simulation frequency must be in the range of frequency specified in the N-port S-parameter file. If the simulation frequency is out of range, an error message appears and the simulation is terminated. You need to either change the simulation frequency or update the N-port S-parameter file to include new frequency data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

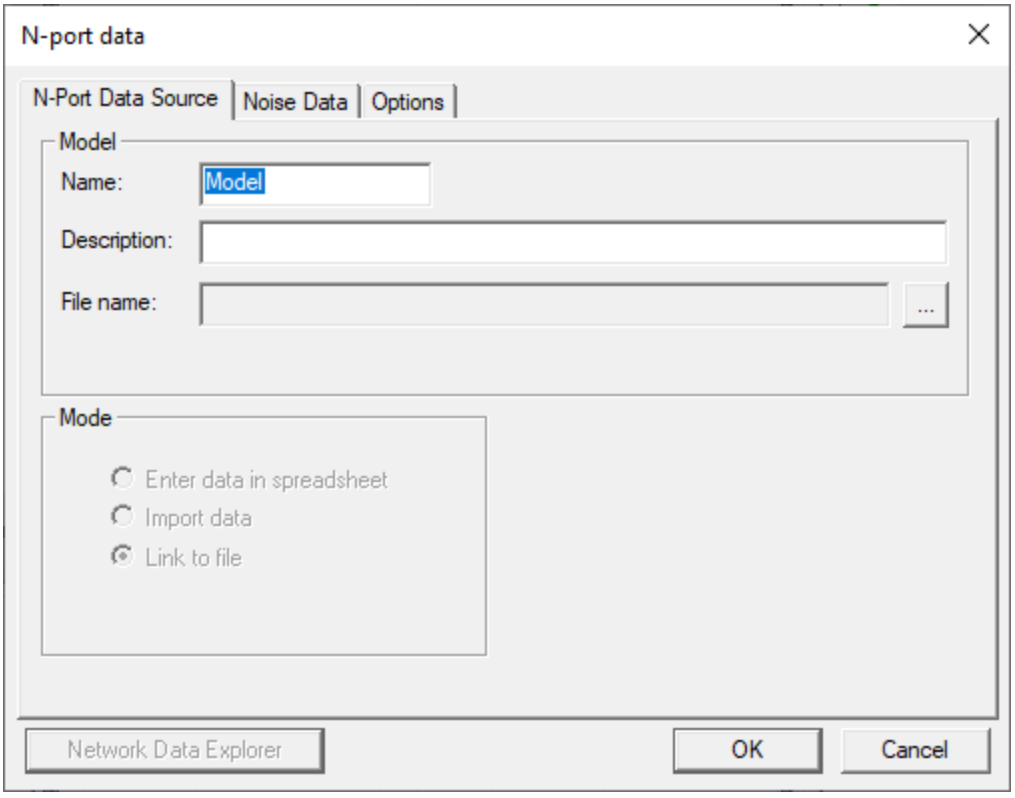
If the simulation frequency does not coincide with any of the N-port frequencies, HFSS approximates the N-port S-parameters at the simulating frequency by interpolating them. The accuracy of the interpolation may deteriorate if the S-parameters contain sharp nulls or peaks. So for N-ports that resonate, ensure there are enough data points in the N-port S-parameters near the resonance.

- N-ports cannot be connected to coupled mode, slotline, or CPW ports.
- If you modify the S-parameter file of an N-port, you must update the N-port in the structure by re-importing the file to use the updated data.

Importing or Linking N-Port Data

If you want to refer to an N-port data file during the solution process, you can import an N-port data file into the active project or link to an N-port data file. The latter method is useful when you want to avoid importing an N-port data file into the project every time it is updated.

1. From the **Project** menu, select **Add Model** > **Add NPort Model** to open the **N-port data** window.



2. Under the **N-Port Data Source** tab, select one of the following data sources:
   - **Import data**
   - **Link to file**

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 one of the following options in the **File** area:
    - **Use Path** if you want to specify the exact location of the N-port data file.
    - **In PersonalLib folder** if the file is located in the program's *PersonalLib* folder.
    - **In userlib folder** if the file is located in the program's *userlib* folder.
    - **In syslib folder** if the file is located in the program's *syslib* folder.
4. Click the **[...]** button and use the explorer window to select the appropriate N-port data file.
5. Click **OK** to return to the **N-Port Data Source** tab of the **N-port data** window
6. Follow the general procedure for inserting N-Port data.

Manually Entering N-Port Data

You have the option to manually enter S-, Y-, or Z-matrix data in the **N-Port Data** window, rather than importing or linking to an existing N-port data file.

1. From the **Project** menu, select **Add Model** > **Add NPort Model** to open the **N-port data** window.
2. Under the **N-Port Data Source** tab, select **Enter data in spreadsheet**.
3. Click the **Network Data** tab and select whether the information being entered is in the **Time** or **Frequency** domain on the **Domain** list.
4. Select the units for the information being entered on the **Units** list.

- Select **S-matrix**, **Y-matrix**, or **Z-matrix**, depending upon the type of data you want to enter.

> **Note:**
>
> If the N-port data file contains complex propagation constant or characteristic impedance values, select **Gamma** or $\mathbf{Z_o}$, respectively, to view the data, but the information is not appropriate to run a simulation.

5. Enter the number of ports to which the N-port is connected in the **Ports** box.
6. Select the format in which you want to enter the data - (magnitude, phase), (real, imaginary), or (dB/, phase) - from the list. Then enter the data for each frequency point in the spreadsheet.
    - Click **Add Frequency** to add information for an additional frequency point.
    - Click **Clone** to create a copy of the selected set of frequency data and place the copy at the end of the spreadsheet. Cloning can save data entry time.
    - Click **Delete** to delete the selected set of frequency data.
7. Follow the general procedure for inserting N-Port data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the Reference Impedance for N-Port Data

If applicable, change the reference impedance for the S-parameter data in an N-port data file. This is done under the **Port Impedance** tab in the **N-port data** window. If applicable, apply the same impedance to all ports or a different impedance to each port.

1. Click the **Port Impedance** tab in the **N-Port Data** window.
2. To apply the same reference impedance to all ports, do the following:
   a. Select **Simple Port Impedance**.
   b. Type the impedance value in ohms.
   c. Select **Admittance** if you want to calculate the admittance matrix, the inverse of the impedance matrix, based on the value entered.
   d. Click **Update**.
3. To apply a different reference impedance to each port, do the following:
   a. Select **Set impedance for each port**.
   b. Select a port from the list.
   c. Select **Impedance** or **Admittance**, then type the real and imaginary parts of the impedance value in the **Real** and **Imaginary** fields.
   d. Click **Update**.

Repeat these steps for each port.

Rotating the Phase of N-Port Data

You can rotate the phase of an N-port's S-parameter data. This is done under the **Network Data** tab in the N-Port Data window.

1. Click the **Network Data** tab in the **N-Port Data** window.
2. Click **Phase Rotation**.
3. If you select **Specify Distance and Effective Dielectric Constant**, enter a distance in the **Length** field and a dielectric constant value in the **Keff** text box.
4. If you select **Time Delay**, enter the delay time in the **Time** field and select a unit in seconds from the list.
5. Click **Update**.
6. Click **OK**.

Phase rotation is calculated for the N-port data and you return to the **Network Data** tab.

Drawing a Connection in the Layout Editor

After inserting an N-port into the project, you must connect it to the model. A new port is created upon connection and is automatically defined as a pad port. The order in which you 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.

ports determines the order of port connection and the solution. So is important to connect the ports in the right order, since most active device S-parameters are not symmetrical.

To draw a connection from an N-port to the model:

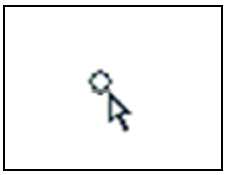
1. Select the edge of the model to which you want to connect the N-port.
2. Select the port on the N-port that you want to connect to the edge.
3. From the **Draw** menu, select **Connection**.

- The edge you selected is connected to the N-port and creates a new internal port upon connection. The N-port data is used during the solution process.
- Ensure to connect all the ports of an N-port to the model.

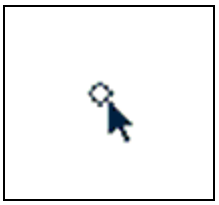
> **Note:**
>
> To remove a connection so that objects can be easily removed and/or reinserted elsewhere, select one or more objects and click **Disconnect from Net** on the **Draw** menu.

Multiple Connections

Additionally, the **Connection** command is an interactive mode that can be used to make multiple connections, one after another. When nothing is selected, and **Draw** > **Connection** is selected, the layout changes to connect mode and the cursor changes to the following:



Then, when the cursor is close enough to select an object that can be connected, the object is graphically highlighted and the cursor changes to the following:

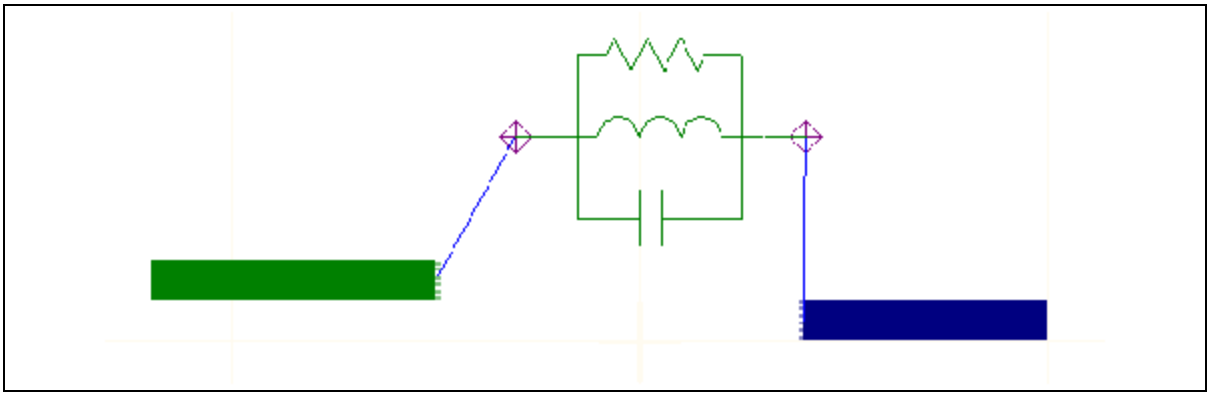


Click on an object to connect. Moving the cursor now extends a feedback segment on the connected object. Click on a second object and the two are connected in the same net.

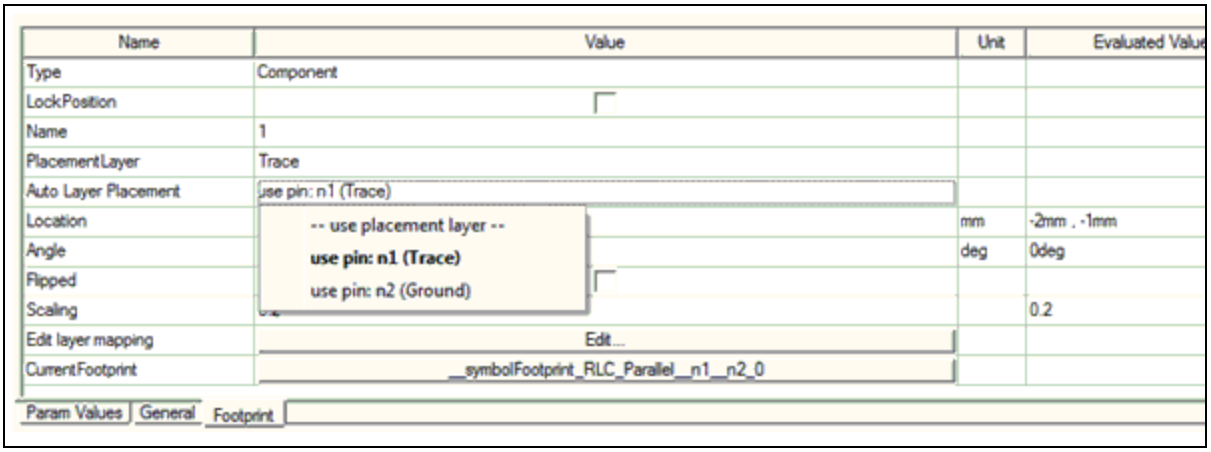
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Components with a Symbol Footprint

Components that do not have footprint geometry are initially rendered on the Symbol layer. When the component is first wired to an object on a physical layer, it renders on that physical layer.



If the component has any pins wired to an object on a physical layer, a new property called **Auto Layer Placement** appears. It is populated with all the pins that are connected to physical layers, plus an additional override option: **use placement layer**.



When the **Auto Layer Placement** is set to **use placement layer**, the **PlacementLayer** property is editable and users may override the layer placement for the component rendering.

Assigning Reference Ports in the Layout Editor

Assign reference ports using the **Draw** > **Port** > **Assign Reference** command. Both selected edges and vias are changed to ports:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Selected edges become edge ports
- Selected vias become pins

Assign reference ports using any of the following scenarios.

Padstack/Padstack

1. Select two padstack instances.

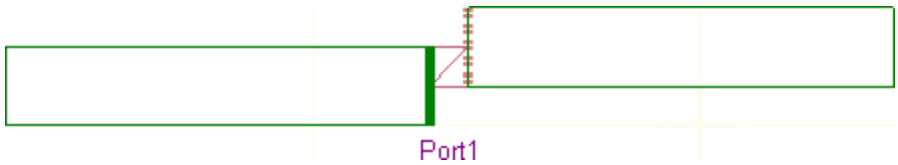
2. Navigate to **Draw** and select **Port** > **Create**

The first padstack selected becomes the port, and the second becomes the reference.

Edge/Edge

1. Select two non-linear edge groups on the same layer.

2. Navigate to **Draw** and select **Port** > **Create**.



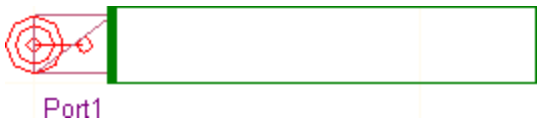
The first edge selected becomes the port, and the second becomes the reference.

Edge/Padstack

1. Select an edge and a padstack.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

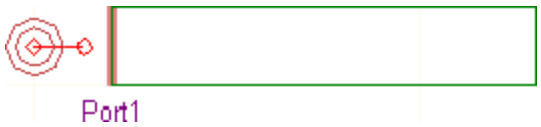
2. Navigate to **Draw** and select **Port** > **Create**.



The first edge becomes the port, and the second becomes the reference.

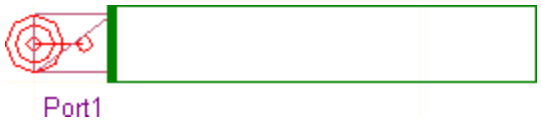
Port/Padstack

1. Select an existing port and a padstack.



OR

2. Navigate to **Draw** and select **Port** > **Assign Reference**



OR

The padstack becomes the reference for the port.

Circuit Ports in Layout

Circuit ports may be created using the following commands:

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 **Draw** menu, select **Port** > **Create Circuit Ports**.
- Right-click in the active area of the **Layout Editor** and select **Port** > **Create Circuit Ports**.

Modes of Operation

- With no active selection, the command activates the circuit port drawing tool.

1. Click at the start point.
2. Click at the end point.
3. Choose the start/end layers.

A circuit port is created between the two points and the tool remains active until it is canceled or until a new tool is selected.

- With an active selection, the command creates a circuit port between the selected edges or pins, or in the case of a single edge, on the edge to the ground.

General Rules

- The start/end points of a circuit port must lie on metal (a negative ground layer is acceptable).
- The HFSS port type can be toggled between gap and circuit if the port geometry represents a valid gap port.

Dielectric Modeling

HFSS and HFSS 3D Layout support finite dielectrics. Dielectric layers are always truncated according to the HFSS Model Extents. Using the extents settings, define how the polygon dielectric and negative signal layers are generated relative to the base polygon. By default, the HFSS Model Extents setting for “Honor primitives on dielectric layers” is selected, but when the check box is unselected, dielectric layers use the extents settings regardless of whether drawn polygons on them. For more information, see HFSS 3D Model Extents.

In contrast, Signal layers can be made of either dielectric or conducting materials and can have internal, user-drawn shapes to create what are sometimes called "dielectric bricks" or "dielectric inclusions”. While sometimes called "bricks", there are no limitations on the dielectric shapes. Signal layers assigned a dielectric material creates objects corresponding to whatever shapes exist on the layer. Signal layers may overlap other Signal layers in elevation, and this provides a great deal of flexibility for representing dielectric object.

From the event that dielectrics on different Signal layers overlap, the Precedence Value on the EM Modeling Properties can be used to resolve ambiguities with lower values having priority. If EM Modeling Properties have not been added or cannot be used to resolve overlaps, precedence is assigned according to material properties.

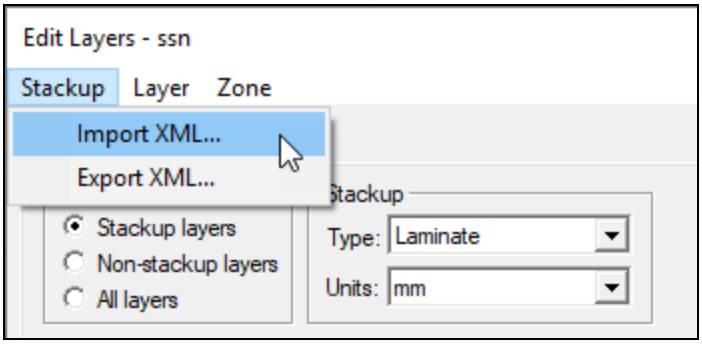
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing a Stackup

In certain instances it may be easier to define your dielectric and signal layer stackup by importing an XML based “control file" using the **Edit Layers** window (e.g., if you have a standard technology process, and you want to assure that all the designs you create use this process, you can import a control file to guarantee that all the stackups are identical).

The XML control file can be created using a third party text editor or a script. In some cases, extremely complex stackups can be more easily created outside of HFSS , then imported, rather than creating the stackups using the HFSS interface. For more information on the format of the XML layer control file, see Layout and ODB++ Control File.

The Edit Layers window has two drop-down menus: **Stackup** and **Layer**.



The **Stackup** menu allows users to import a stackup from an XML file and export the stackup in the window to an XML file.

To import a stackup, open the **EditLayers** window by clicking its toolbar icon or by selecting **Layout > Layers**. Next, click on **Stackup** > **Import XML**. A series of window opens that allow you to browse to the XML control file and import the stackup. The exact series of steps depends on whether the design has existing layers and whether the layers contain geometry. In certain instances the import mechanism adds additional dielectric/signal layers to create the pancake stackup.

The Design Does Not Have Existing Layers

The Layout/ODB++ Control file can be imported directly. A **FileOpen** window appears that allows you to browse to the XML layer control file.

The Design Has Existing Layers

After clicking **ImportStackup**, a warning may display that indicates at least one of the existing layers contains geometry. A window then offers the option of replacing or updating the existing layers. After choosing to update or replace existing layers, a **FileOpen** window appears that allows you to browse to the XML layer control file.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Replace existing layers** – All existing layers in the design are discarded and layers on the Layout/ODB++ Control File are inserted into the design. Geometry on any of the existing layers may be moved to different layers. After importing a stackup into a design with existing geometry, it is important to recheck your design.
- **Update existing layers** – The import mechanism attempts to merge the existing design layers with those in the Layout/ODB++ Control File. In some cases the import mechanism cannot merge the layers and you are not able to import the Layout/ODB++ Control file. The following examples illustrate the results when a one-to-one match does not exist between existing and imported layers. The XML file and existing stackup contain layers of type signal ("S") and dielectric ("D"). The name "S1" signifies a signal layer, but the layer names you select can be arbitrary.

Example 1: One to one mapping. This is the simplest case where existing layers are updated with new material properties and thicknesses. The matching of layer names is case insensitive.

XML File	Existing	Resulting Stackup
S1	S1	S1
D0	D0	D0
S0	S0	S0

Example 2: Merge not possible. In this case the layers in the XML file are very different on the layers in the existing stackup and the import mechanism cannot create a merged stackup.

XML File	Existing	Resulting Stackup
S1	S3	Not possible to merge
D0	D1	
S0	S2	

XML File	Existing	Resulting Stackup
S1	S1	Not possible to merge
D0	D1	
S0	S0	

Example 3: Additional layers. In this case the XML file has one or more additional layers. The import mechanism updates existing material properties and thicknesses and add the additional layers.

XML	Existing	Resulting

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File		Stackup
D1		D1
S1	S1	S1
D0	D0	D0
S0	S0	S0

Example 4: Automatic layer addition. The import mechanism adds dielectric or signal layers to create a strict pancake stackup.

XML File	Existing	Resulting Stackup
D1		D1
S2		S2
D0		D0
S1	S1	S1
S0	S0	S0_DIEL
		S0

Layer "S0_DIEL" is a dielectric layer added by the import mechanism with defaults for the material and thickness.

Components

A component (i.e., a component instance) is a group of objects (e.g., pins, vias, primitives, et cetera.) on a layer that constitute a chip or RLC. When a component is created, a definition for it is created in the **Project Tree** > **Definitions** > **Components** folder.

From the **Layout Editor**, the borders of the component appear as a box that encloses all the component's members. **Click+drag** the bounding box to move the component. Individual members of the component cannot be moved individually. Once a component is created, the positions of its members are frozen in relation to one another.

You can add or remove entities from components. When you do, the component's border shifts accordingly. The component's definition can change because component models are partially defined by the number of pins they contain. If the number of pins changes, then a new definition is created.

To highlight a component, either select the reference designation of a component in the **Components** window or **click+drag** the cursor over the components.

When a component's bounding box is highlighted, all members of the component are also highlighted. If an entity in the component is not highlighted, then that entity is not a member of the component.

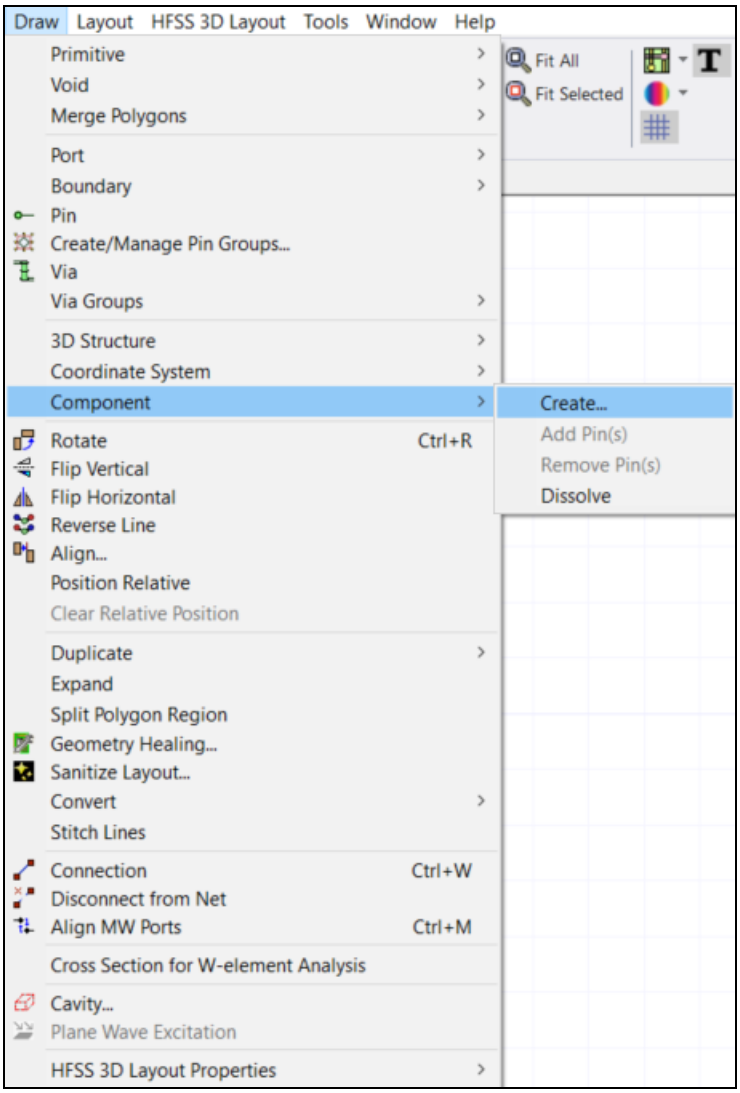
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An active component can be moved within a hierarchical design to a higher or lower position within the global Coordinate System (CS). Refer to Component Placement in HFSS 3D Layout.

Creating a Component

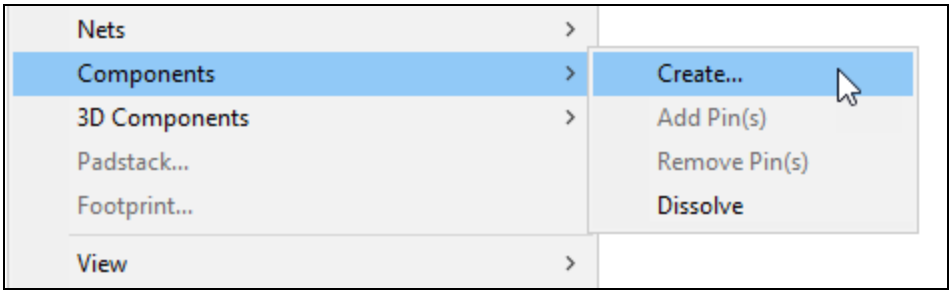
You can create a component from pins, vias, and primitives. A bounding box shows the border of the component unless another component is a member of the new component. Follow these steps to join components.

1. From the **Layout Editor**, select the components.
2. Perform either of these steps to open the **Create Component** window.
    a. From the **Draw** menu, select **Component** > **Create**.

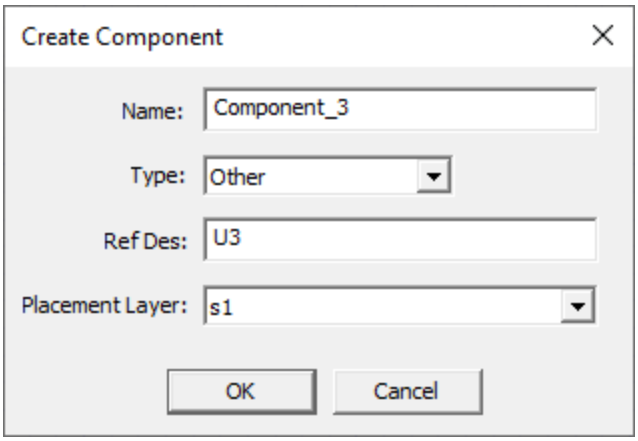


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. From the **Layout Editor**, right-click and select **Components** > **Create** .



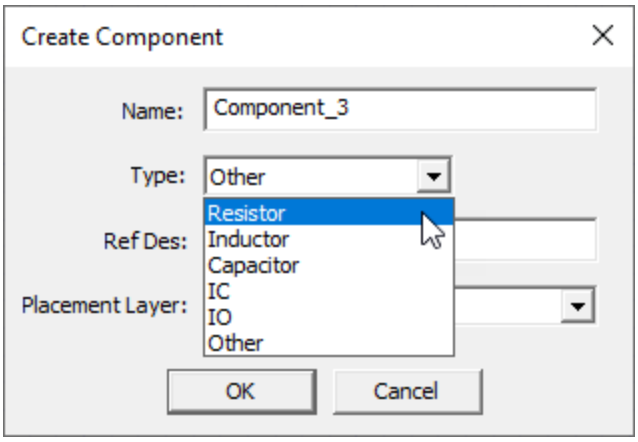
The **Create Component** window opens.



3. Once the **Create Component** window opens, enter the following information into the fields and 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.

- Name: — enter a name for the new component
- Type: — choose **Resistor**, **Inductor**, **Capacitor**, **IO**, **IC**, or **Other**.



> **Note:**
>
> The type you choose determines the component's model type.

- Ref Des: — enter an alternative reference destination to identify this component in the active design

> **Note:**
>
> This is optional. A reference destination is automatically supplied.

- Placement Layer: — select which stackup layer the component should be on

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 new component, overlapping pins are automatically filtered out (i.e., ignored). This prevents downstream issues involving pin groups, terminal creation, meshing, solving et cetera. The following rules determine which pin is retained when overlaps occur:

- If one pin is multilayer and the other is present only on a single layer, the multilayer pin is preferred.
- If both pins are multilayer, the alphabetically lower pin name is preferred.
- If both pins are single-layer, the larger pin (by size) is preferred.

If overlapping pins are detected during area selection, the ignored pins are listed in the **Message Manager** window.

4. Click **OK**. The new component is listed in the dockable **Components** window.

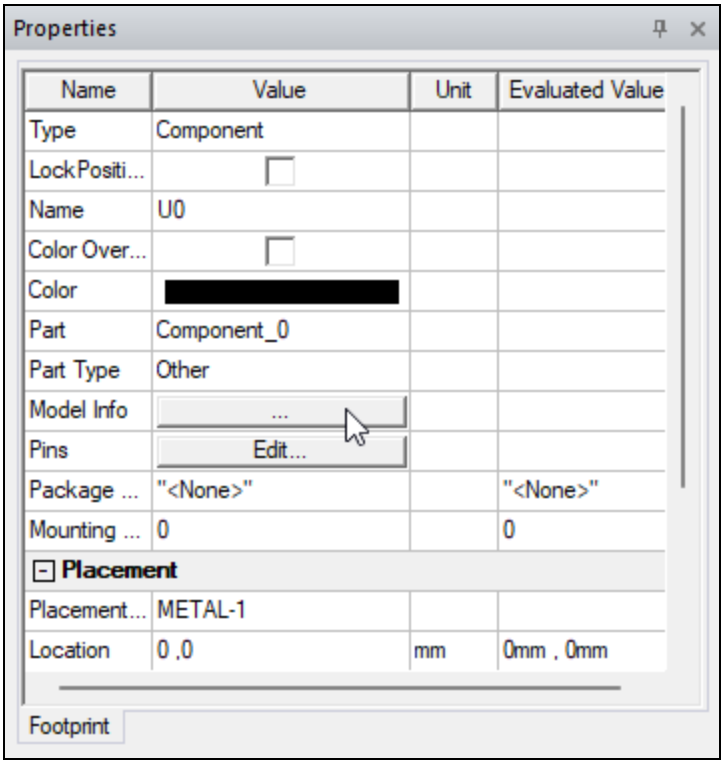
The process is complete.

Editing a Component

A component is a collection of entities grouped together with a common set of configurations. Use components to configure HFSS 3D Layout objects. Follow these steps to create and edit these configurations in the **Set Selected Component Models** window.

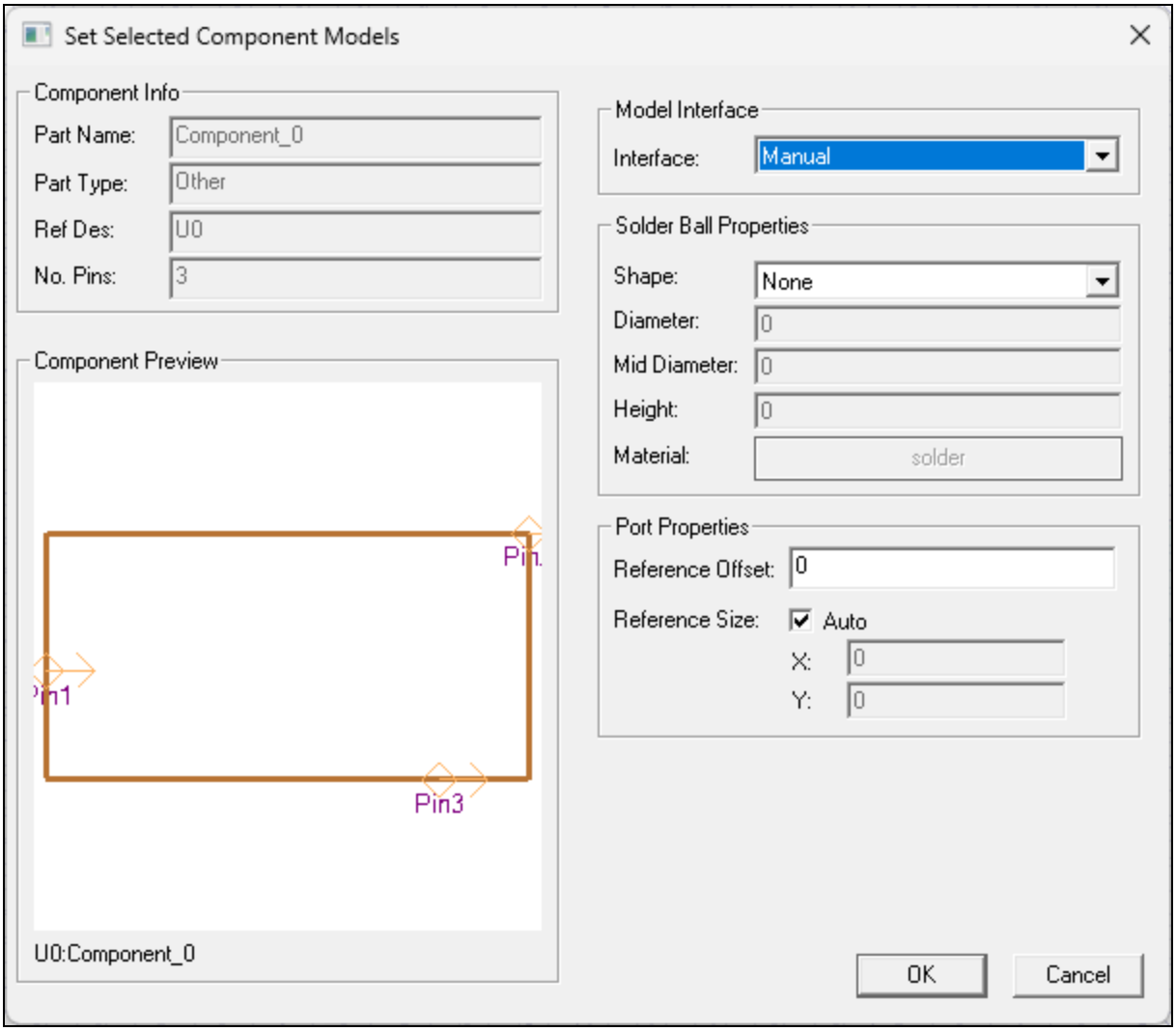
1. From the **Layout Editor**, select a component to populate the **Properties** window.
2. Click the **Mode Info** button to open the **Set Selected Component Models** window.

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 **Set Selected Component Models** window, view information about the selected component. The exact configurations depend on the model type (e.g., an assigned electrical model can characterize how a passive device behaves).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



\

The **Part Type** determines what model type a component can have:

Part Type	Model Type
Resistor Inductor Capacitor	• RLC network: assign a simple parallel or series RLC network. • S-Parameter model: create a model from a touchstone

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part Type	Model Type
	file. • Circuit model: create a model in the **Schematic Editor**. This is also the model type for legacy netlist models. • Library: select a model on the installed vendor libraries. • SPICE model: click **Import SPICE Model** to attach a SPICE model.
IC (integrated circuit) IO (input output) Other	• Manual: Create a custom configuration. • SPICE model: click **Import SPICE Model** to attach a SPICE model.

For S-Parameter and Library models, users can specify a reference net (e.g., a ground/power net), to use as reference for the terminals of the components. Whether to provide a reference net is entirely at user discretion. If no reference net is provided, one will be automatically determined.

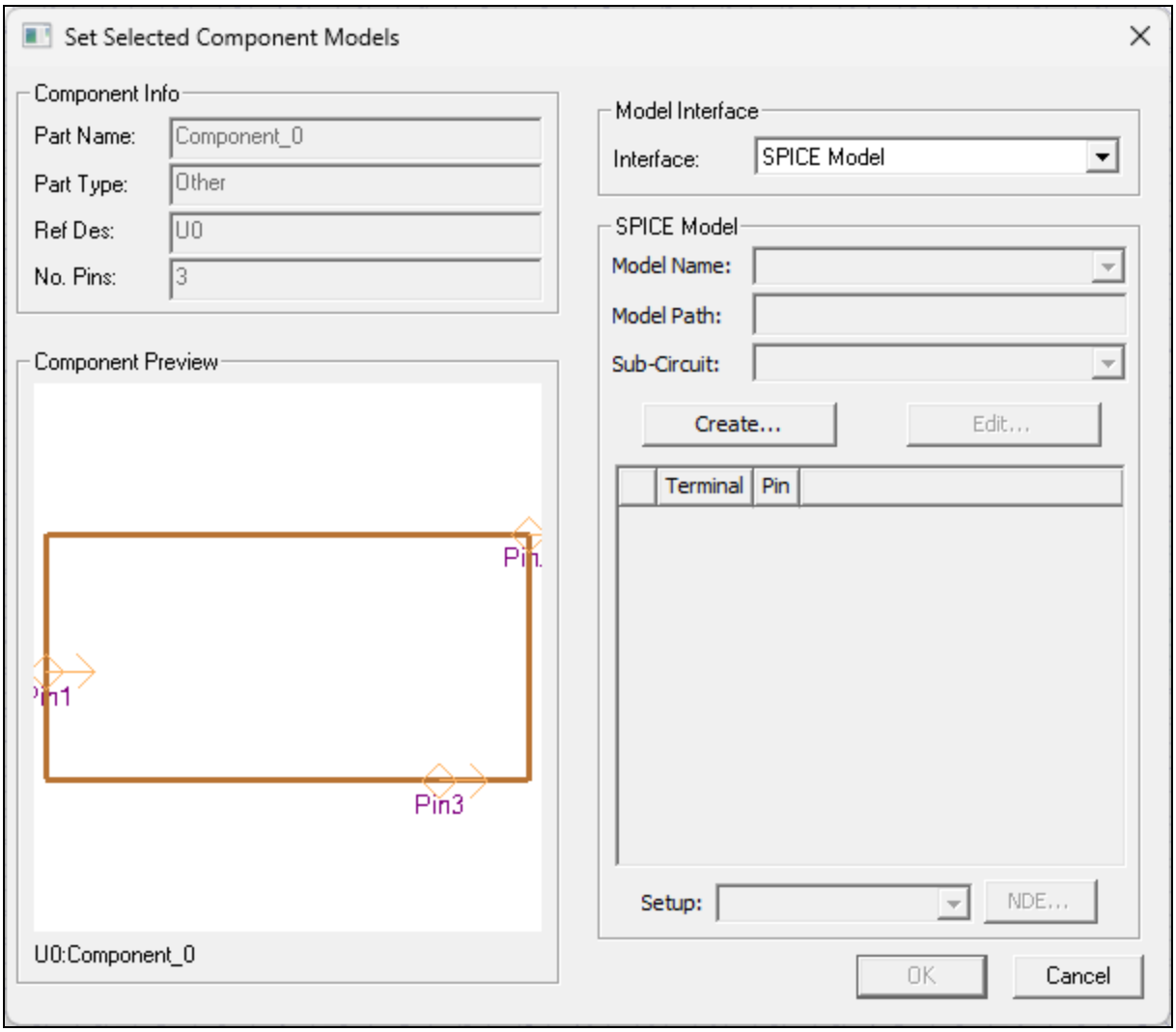
Circuit models supersede netlist models. Continue to edit any netlist model that exists in a legacy project or create a new circuit model with the **Schematic Editor**. Netlists cannot be edited in or converted to a format that can be edited in the **Schematic Editor**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SPICE Model

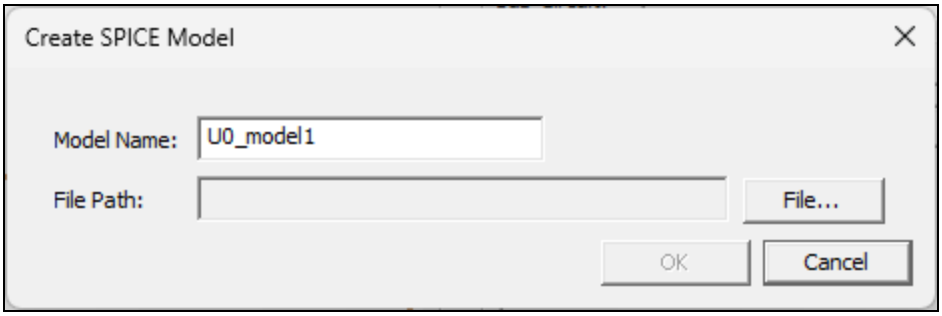
From the **Model Interface** drop-down menu, select **SPICE Model** to change the visible options. Then follow these steps to attach a SPICE model to a specific component.

1. If a SPICE model has been prepared, select it from the **Model Name** drop-down menu.

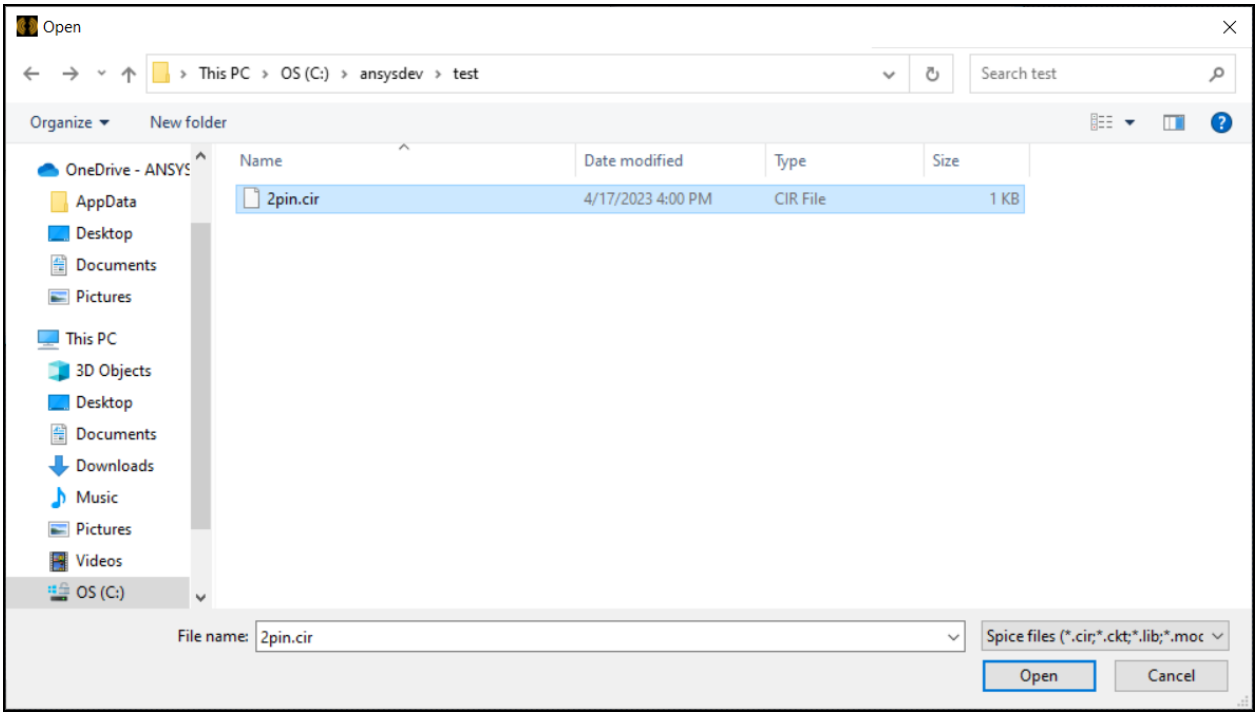


2. Click **Create** to open the **Create SPICE Model** window.
3. Enter a **Model Name** or accept the default (e.g., **U0_model1**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

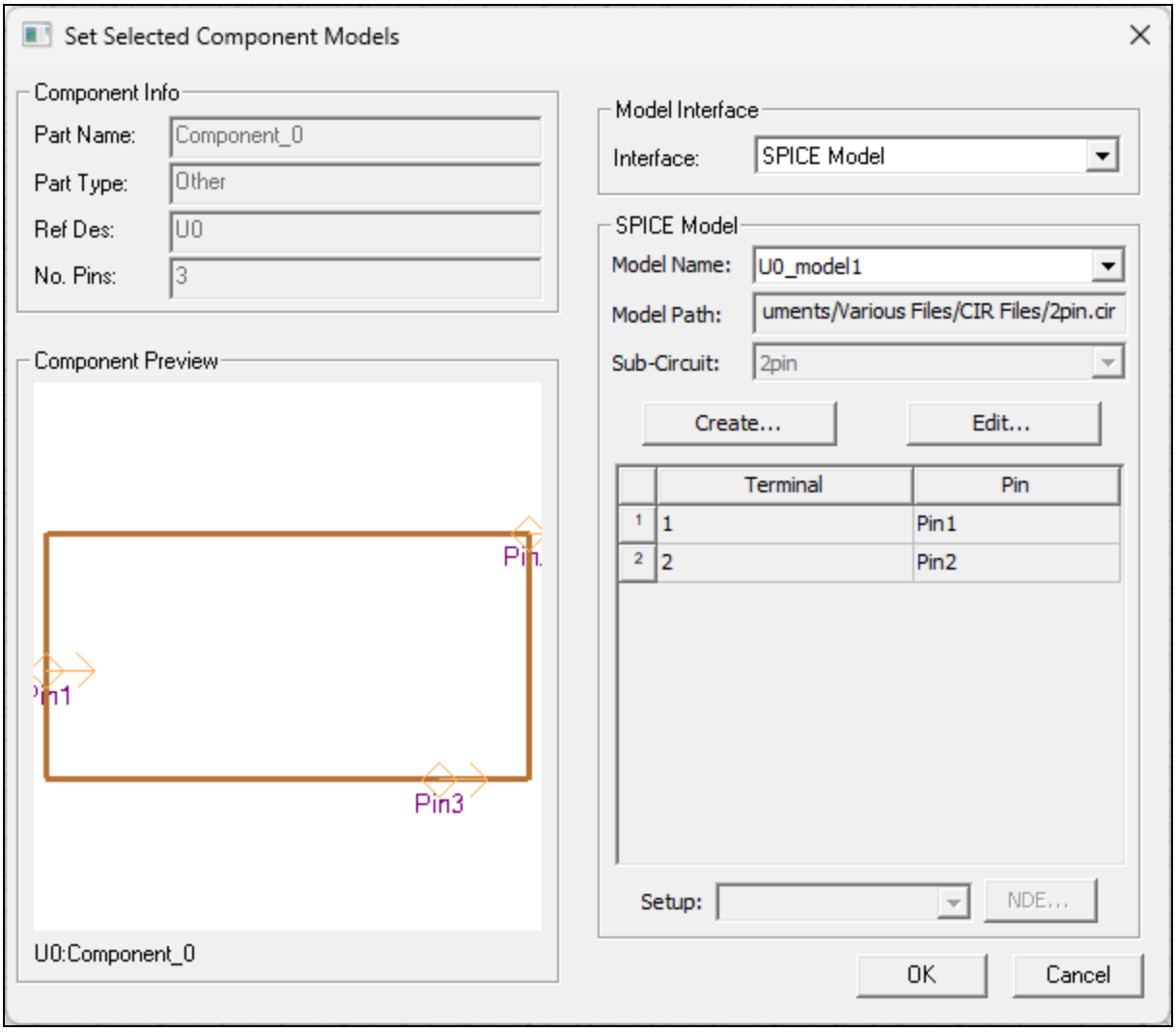


4. Select **File** to open an explorer window.



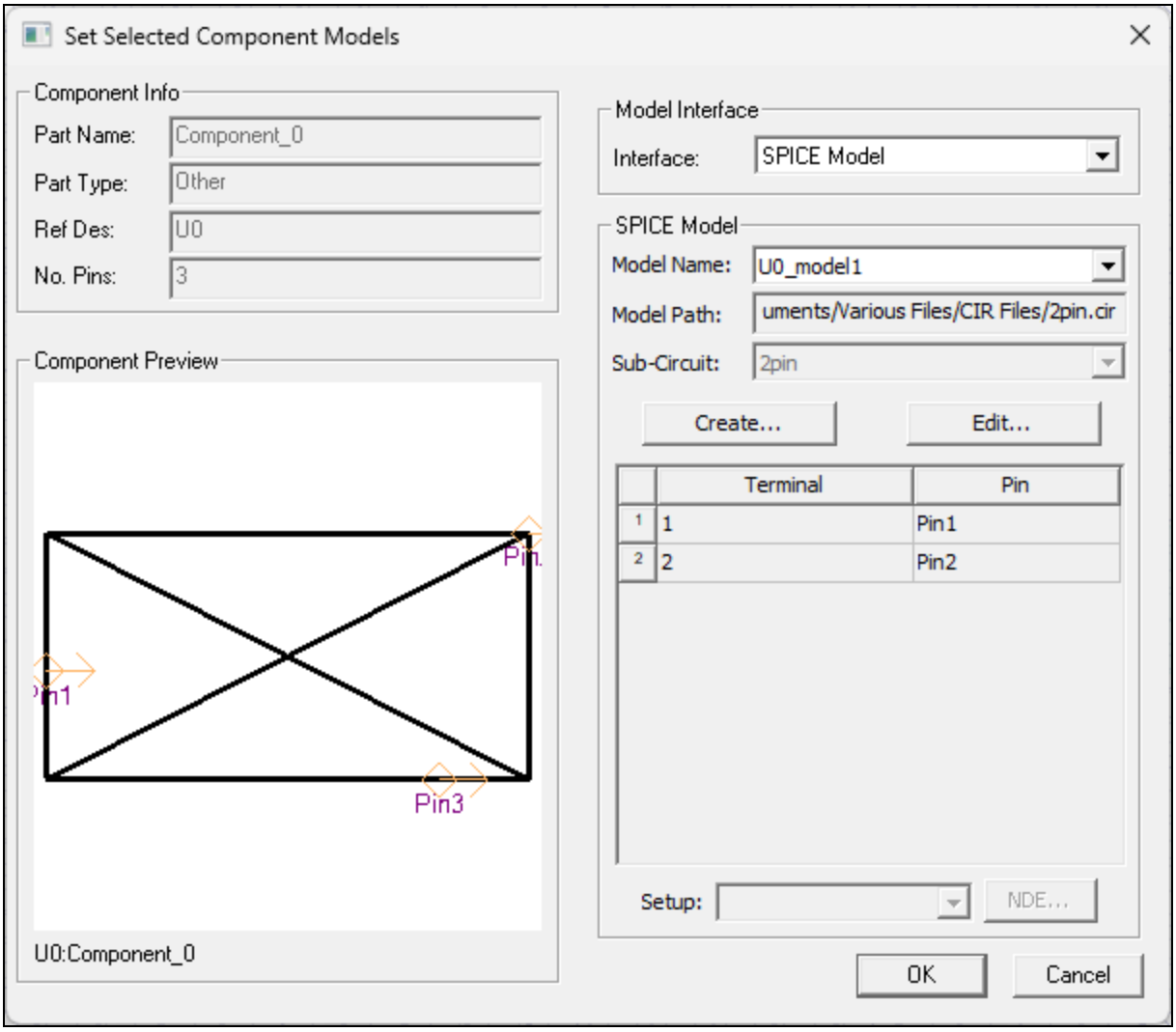
5. Navigate to and select a **Spice file** (e.g., **2pin.cir**). Then click **Open** to close to the explorer window and return to the **Set Selected Component Models** window, which is now populated with the SPICE file's parameters. This includes **Model Path**, active **Sub-Circuit** (if any), and the current **Terminal/Pin Mapping**.

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 available, select a **Sub-Circuit** from the drop-down menu.
7. Make adjustments to **Terminal/Pin Mapping** by clicking in the **Pin** column cells and selecting a pin from the drop-down menu. Repeat, as appropriate.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

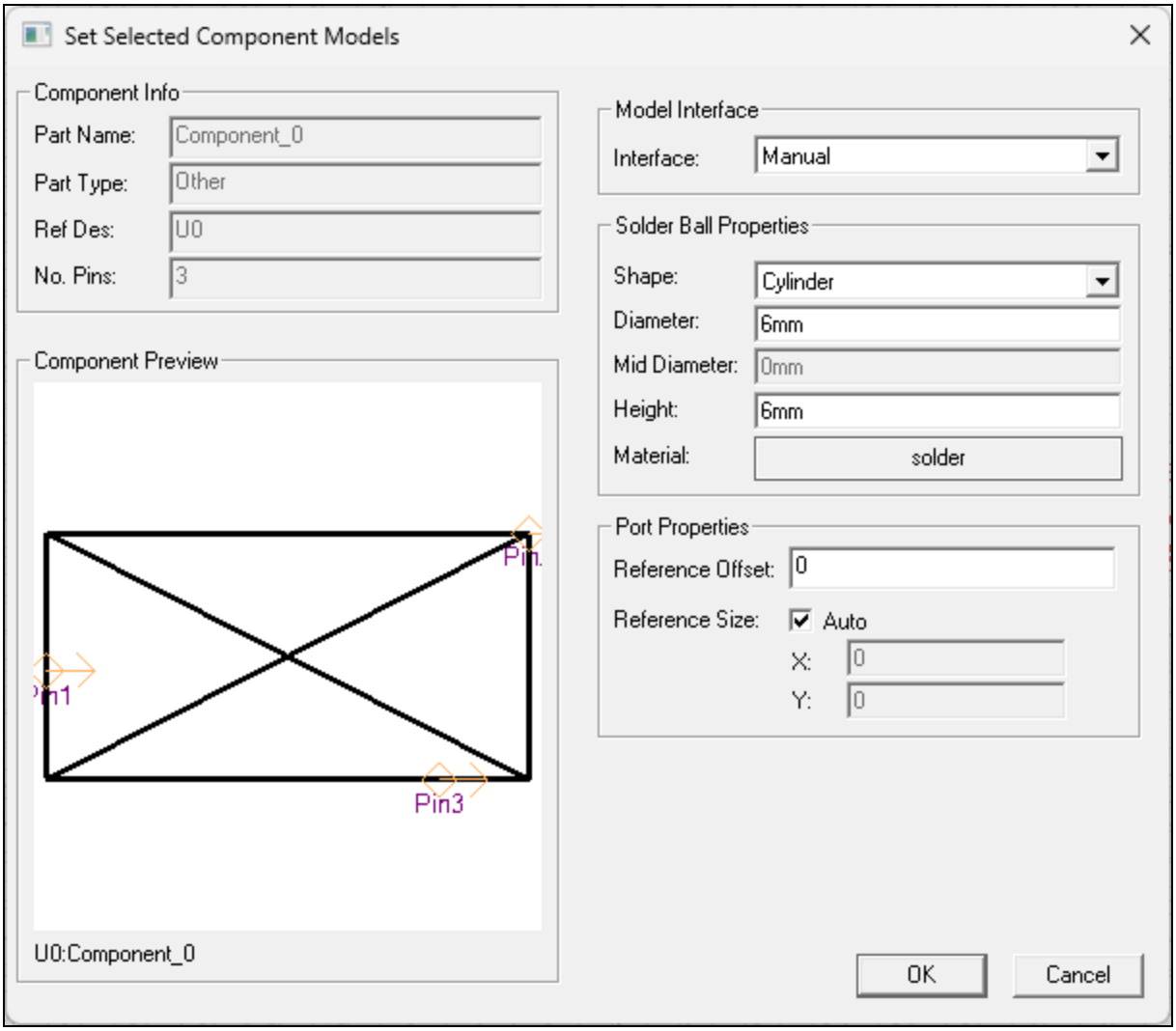


Manual Model

A **Manual** model type contains die, solder ball, and port properties. These properties apply across the component. So, if you configure the **Solder Ball Properties**, every pin on the component is configured with the same type solder ball, positioned uniformly above or below, and generates a common reference for HFSS simulation.

> **Note:**
>
> **Solder Ball** and **Port Properties** are common for IC, IO, and other components. **Die Properties** are only available for IC components.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Die Properties

IC components's die properties determine the z-orientation and connections to the component's pins on its stackup layer. There are three types of die properties: **None**, **Flip chip**, and **Wire bond**.

If the type is **None**, there is nothing connecting to the pins and no configurations to set.

With **Flip chip**, configure the component's pins to the top or bottom side of its layer. **Chip up** places the pins on the bottom side of the layer, while **Chip down** places them on the top side. The pins' default depends on where in the stackup the component is.

When the **Flip chip** controls are available, the component's pins connect via solder balls. So the **Solder Ball Properties** become available.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wire bond type components have all pins in the same orientation, either on the top (**Chip up**) or bottom (**Chip down**) side of the layer. Set the **Height** of the wires as measured on the layer.

Solder Ball Properties

When available, solder balls are identical across a component. They have the same diameters, height, and material. Solder balls can be spheroids or cylinders. Cylinders have a uniform diameter along their height.

Spheroid solder balls are spheres with the top and bottom truncated. Their **Diameter** is the diameter of the interface at the pin and whatever it is connecting to. The **Mid Diameter** is the diameter of the ball at its midpoint.

If **Shape** is set to **None**, the component does not have solder balls.

> **Note:**
>
> When **Die Properties** are available, solder balls can only be configured for **Flip Chip**.

Port Properties

Solder balls and wire bonds are drawn using a reference plain that is controlled by the **Port Properties**. By default, this plain is as large as the component and sits at the top of the solder balls and wire bonds.

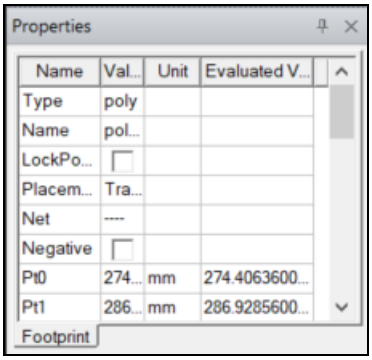
The solder balls can have a **Reference offset** which moves the reference plane a configurable distance on the top of the solder balls.

The reference plane can be resized for solder balls and wire bonds. If **Auto** is enabled, the plane is the size of the component. If unavailable, set the dimension of the plane on the x- and y-axis. The plane can be as large or as small as selected.

Creating or Editing Package Definitions for Components

ODB++ and IPC2581 imports may include properties in their IDF files that describe the component outline or height properties of a component, while IDF imports may bring in thermal properties of a component. These properties describe the component's materials, thermal, power, outline, and heatsink characteristics. These properties can also be added to new components and are accessible through the **Footprint** 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.

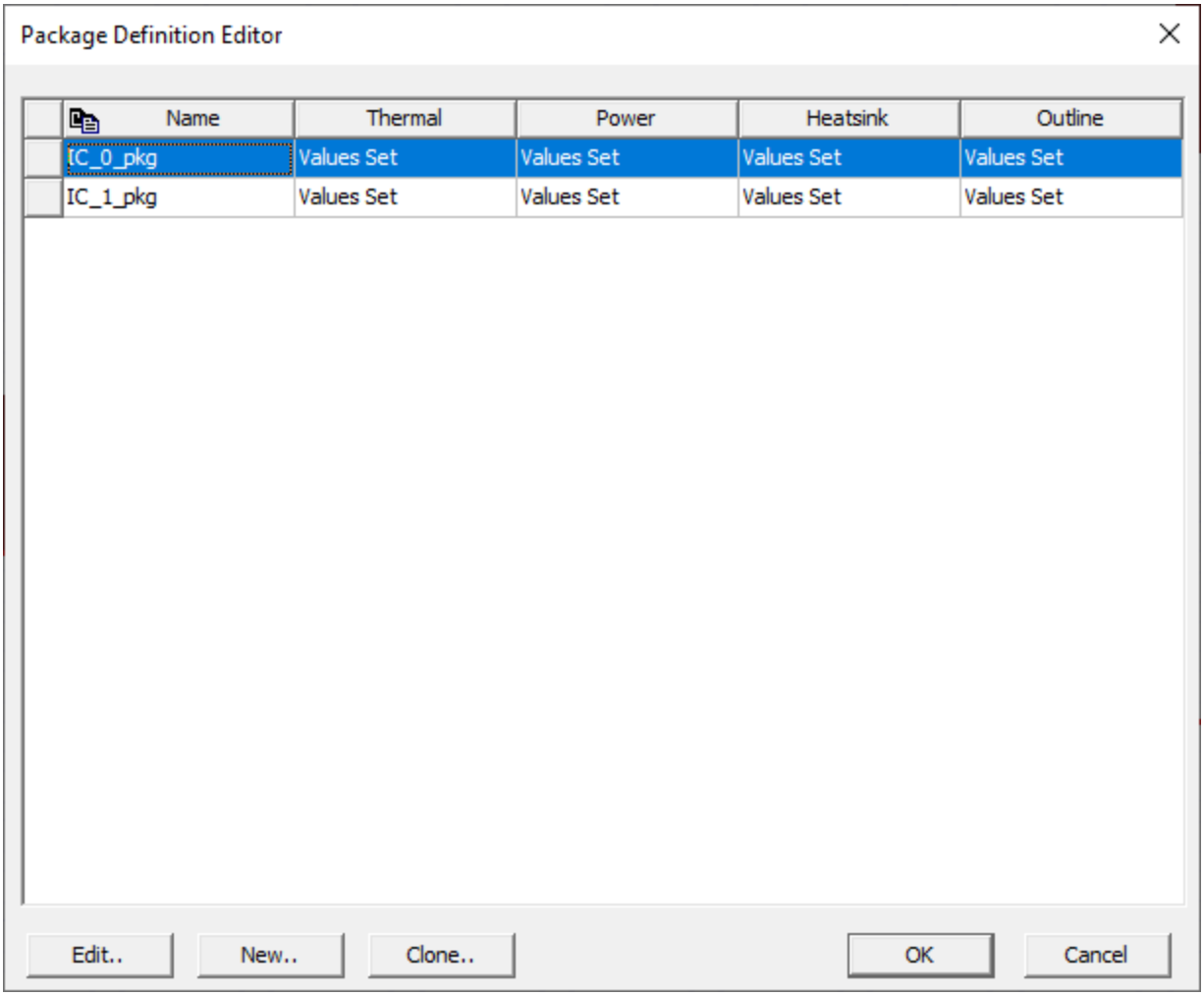


> **Note:**
>
> Package definitions are only used for Icepak thermal analysis with linked PCB components, and when IcepakFEA projects import an Electronics Database (EDB) directly. It is not used by the electromagnetic solvers.

Follow these steps to add, create, or edit a new or existing package to a component.

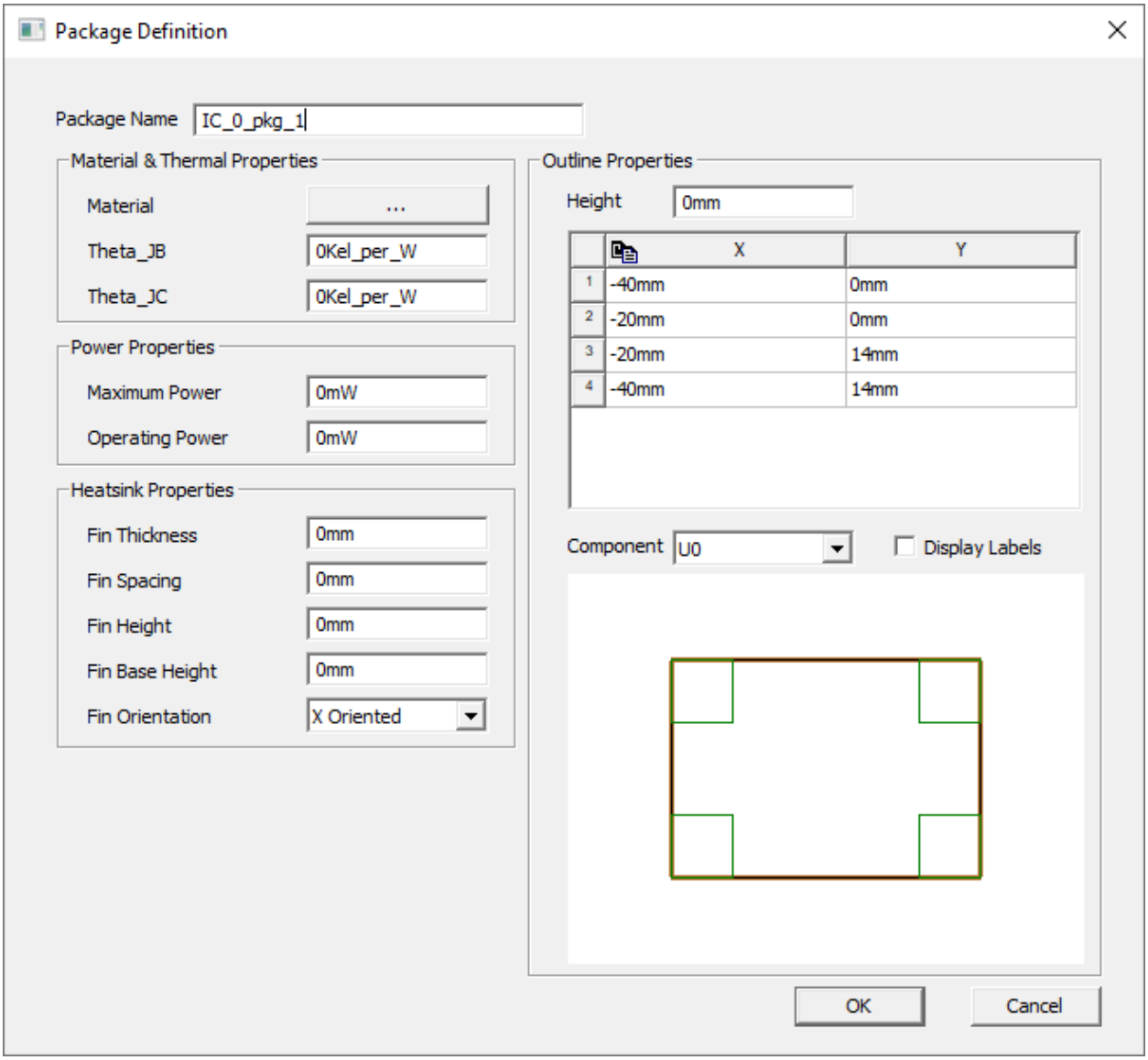
1. From the **Components** window, click a component.
2. From the **Footprint** tab of the **Properties** window, click in the **Value** cell of the **Package Definition** row and select **Edit** to open the **Package Definition Editor**. It lists all packages available in the design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. To add an existing package, select the package from the list and click **OK**.
4. To create a new package, click **New**.
5. To edit an existing package, select the package from the list and click **Edit** to open the **Package 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.



6. Adjust the properties as selected.
7. Click **OK**.
8. In the **Package Definition Editor**, select a package from the list and click **OK**.
9. To view the package outline, enable **Display package graphics** in the Layout Options.

Packages contain four types of properties: materials and thermal, power, heatsink, and outline. The **Materials & Thermal Properties** are the materials the heatsink is made of and measured properties that describe the rate of its heat transfer from a junction to another location. Specifically they are:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Materials**: Allows you to select materials.
- **Theta_JB**: thermal resistance from junction to board. Include the unit Kel_per_W (Kelvin per Watt.)
- **Theta_JC**: thermal resistance between the junction and a point on the outside surface of the package (i.e., the case). (See the IDF specification 3.0 for details.) Include the unit Kel_per_W (Kelvin per Watt.)

The **Power Properties** are:

- **Maximum Power**: the greatest amount of power the package can handle.
- **Operating Power:** the average amount of power experienced during a typical load.

The **Heatsink Properties** simulate a three-dimensional modeling component that can exchange radiation with other objects in the model. The package defines an extruded heatsink which consists of a base and fins that run parallel to each other across the base.

HFSS 3D Layout produces as many fins as fit given the settings of these properties:

- **Fin Thickness**: how wide each fin is.
- **Fin Spacing**: how far apart each fin should be from its nearest neighbor.
- **Fin Height**: how tall each fin is.
- **Fin Base Height**: how tall the base on which the fins sit is.
- **Fin Orientation**: whether the fins are oriented along the X or Y axis.

The **Outline Properties** are a 3D representation of the component. The table lists the X and Y coordinates for the corners of the component and the **Height** is the elevation of the component box along the Z axis. Specify which component is represented in the **Component** drop down list. Click **Display Labels** to label the pins and vias on the diagram.

Intersection Checks for 3D Components

Intersecting 3D components (e.g., 3D structures that partially or completely overlap) can cause a validation check to fail. If the validation check fails, an error message will appear in the **Message Manager** window. The error message may provide detailed feedback about which objects are intersecting. The error message varies, depending on whether the components are unencrypted, encrypted but visible, or encrypted and hidden. If applicable, users can deactivate the intersection check conducted during validation (continue to "Deactivating the Intersection Check" on page 7-285).

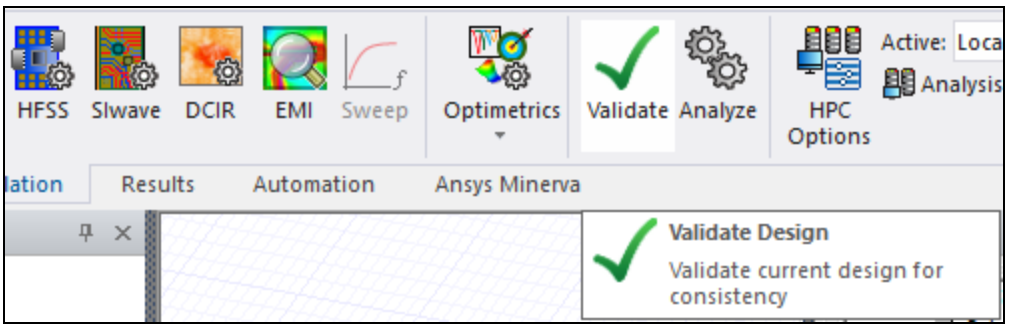
Follow these steps to attempt validation with intersecting components, then view the error message and details (if any).

1. If users have not already done so, they can add a setup to the design and analyze the setup (refer to HFSS 3D Layout 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.

2. To open the **Validation Check** window, do one of the following:

   - Navigate to the **Simulation** tab and select **Validate** from the **Simulation** ribbon.

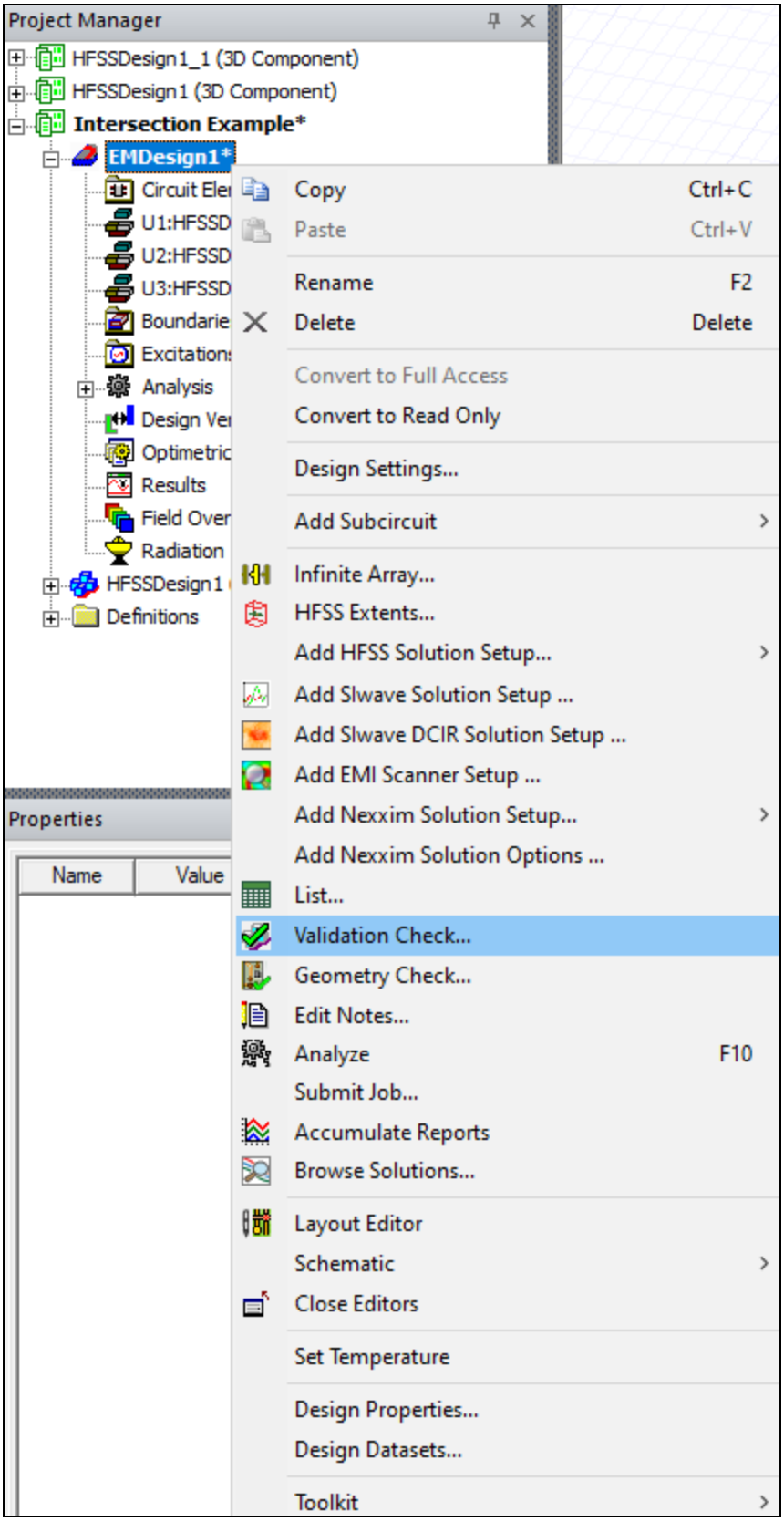


   - From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]**. Then right-click the active design (e.g., **EMDesign1**) and select **Validation**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

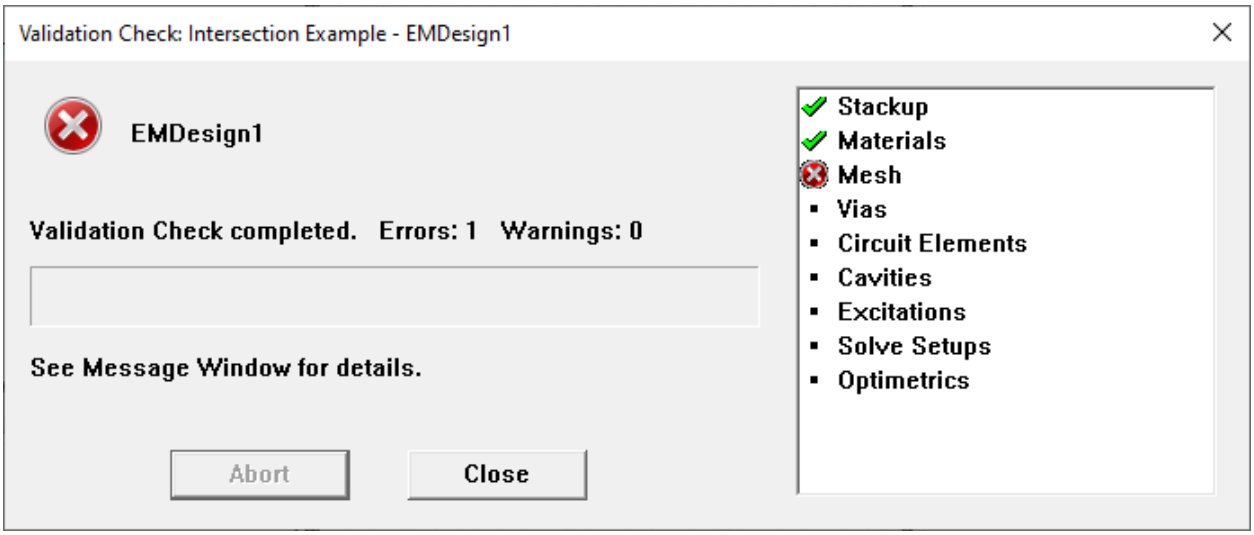
Check.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

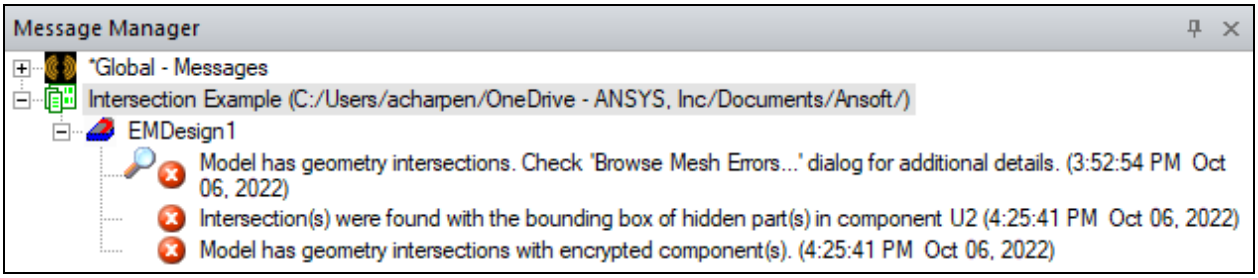


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The **Validation Check** window will display errors/warnings as they occur. If the design includes intersecting objects, the window will display a mesh error. Click **OK** to close the **Validation Check** window.

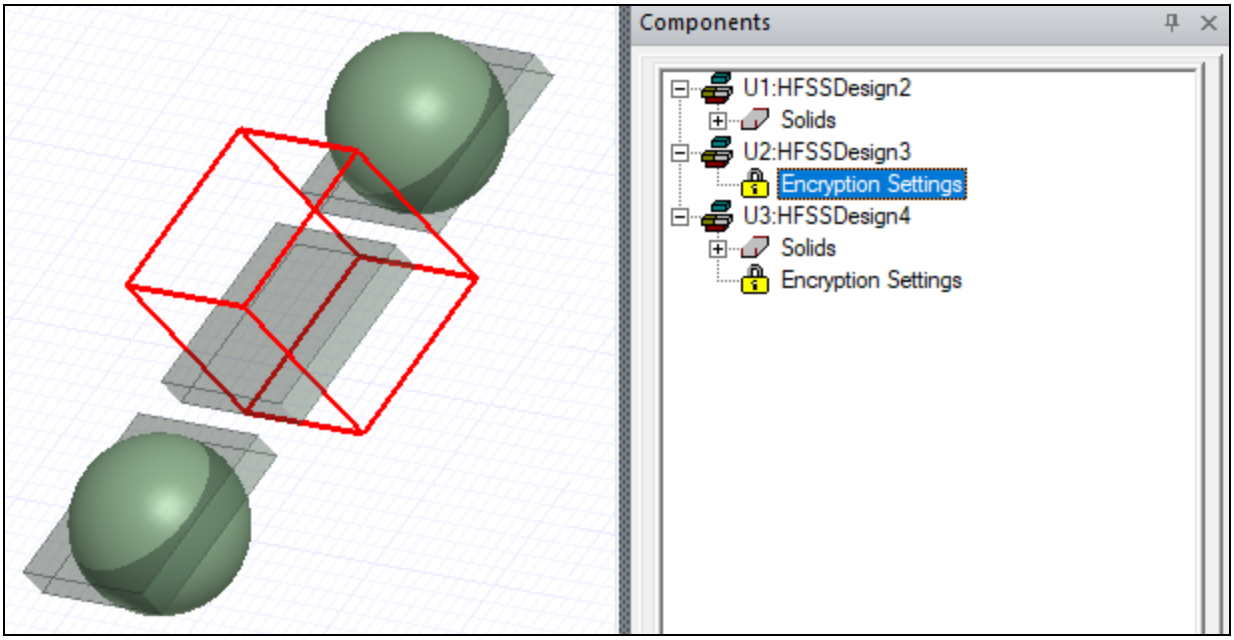


4. Navigate to the **Message Manager** window and identify the mesh error message. Depending on the level of encryption (e.g., unencrypted, encrypted but visible, and/or encrypted and hidden), the specific mesh error(s) will vary.

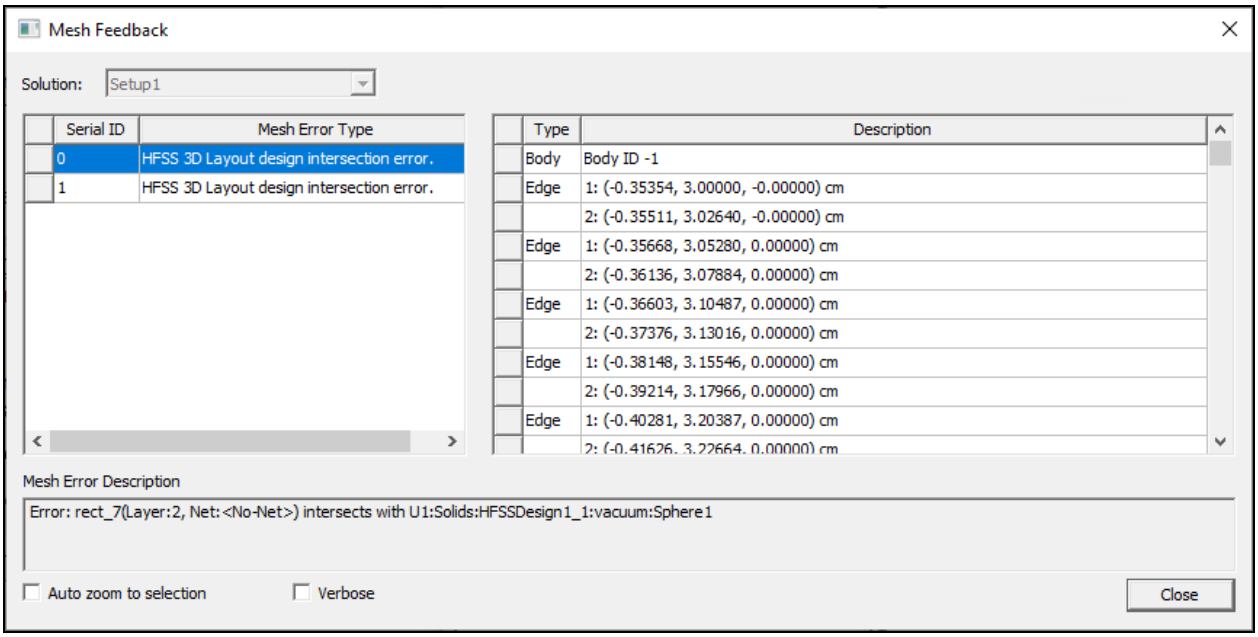


Encrypted, unencrypted, and hidden encrypted components can potentially exist within the same design. Refer to Importing and Exporting Components for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click the mesh error(s) for unencrypted or encrypted but visible components to view a detailed list of the intersection points, click the error message to open the **Mesh Feedback** 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 a Mesh Region to Validate Encrypted Designs

If Electronic Desktop identifies an intersection with an encrypted and hidden component, it is impossible to ascertain the exact coordinates of the intersection or whether the intersection invalidates the design since the exact parameters of the hidden component are unknown (i.e., only its bounding box is calculated).

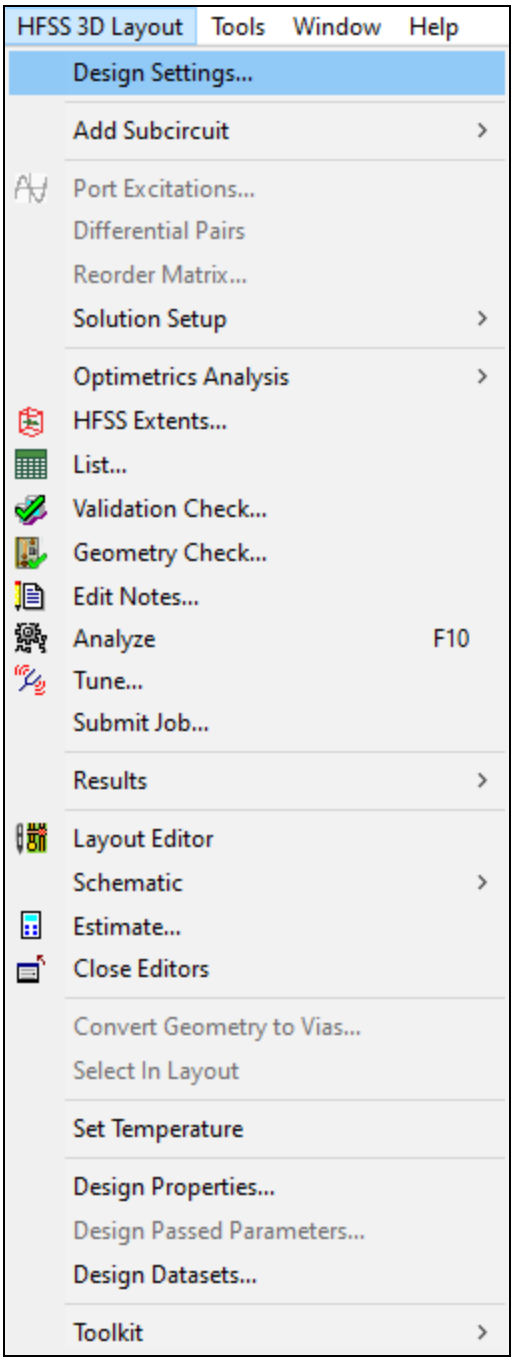
Ansys suggests creating a mesh region around the component. Mesh generation will automatically cease if the generator encounters an intersection. If mesh generation is successful, then the intersection is valid. Refer to "Meshing In HFSS 3D Layout " on page 17-1 for more information.

Deactivating the Intersection Check

If the components in the design are all unencrypted, users can deactivate the intersection check altogether. Deactivating the check has no impact when validating a design that includes an encrypted component. Follow these steps to deactivate the intersection check.

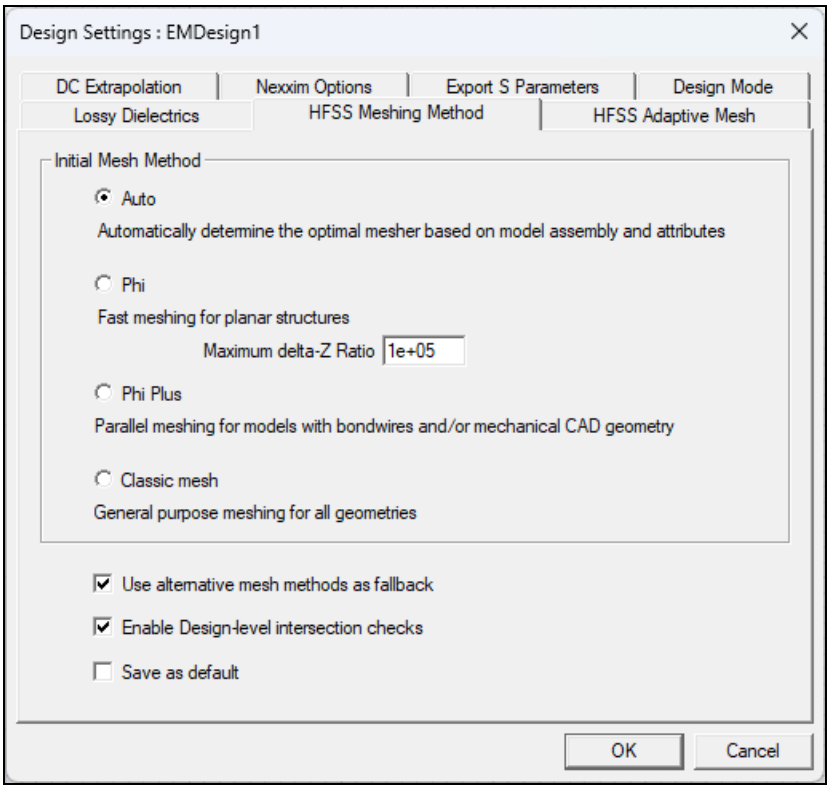
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From **HFSS 3D Layout**, select **Design Settings** to open the **Design Settings** window.



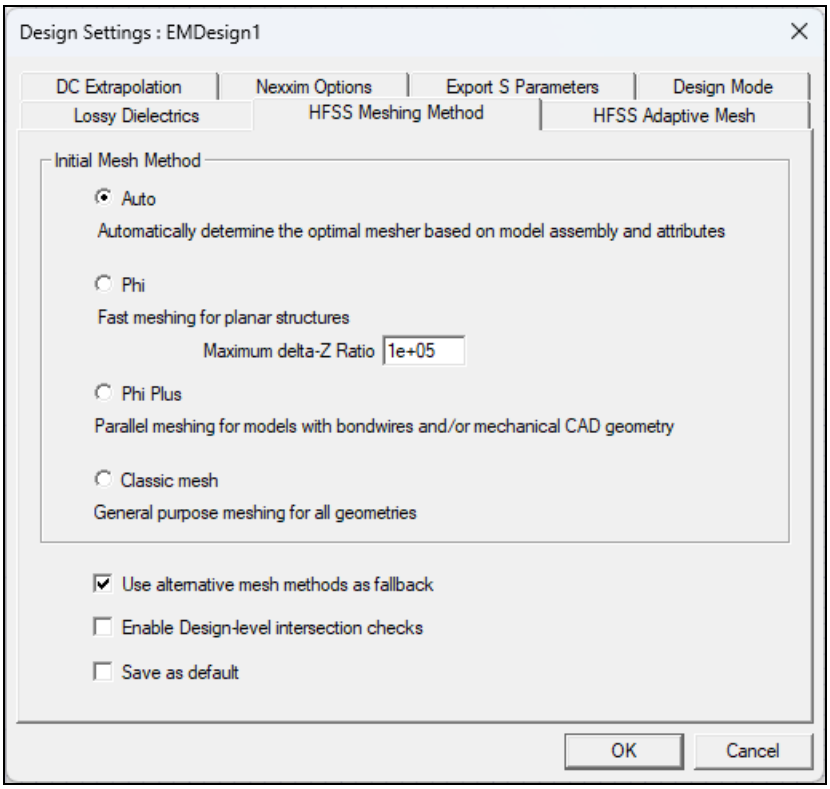
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Navigate to the **HFSS Meshing Method** 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. Remove the check from the **Enable Design-level intersection checks** box.



4. Click **OK** to close the **Design Settings** window.

Next Steps

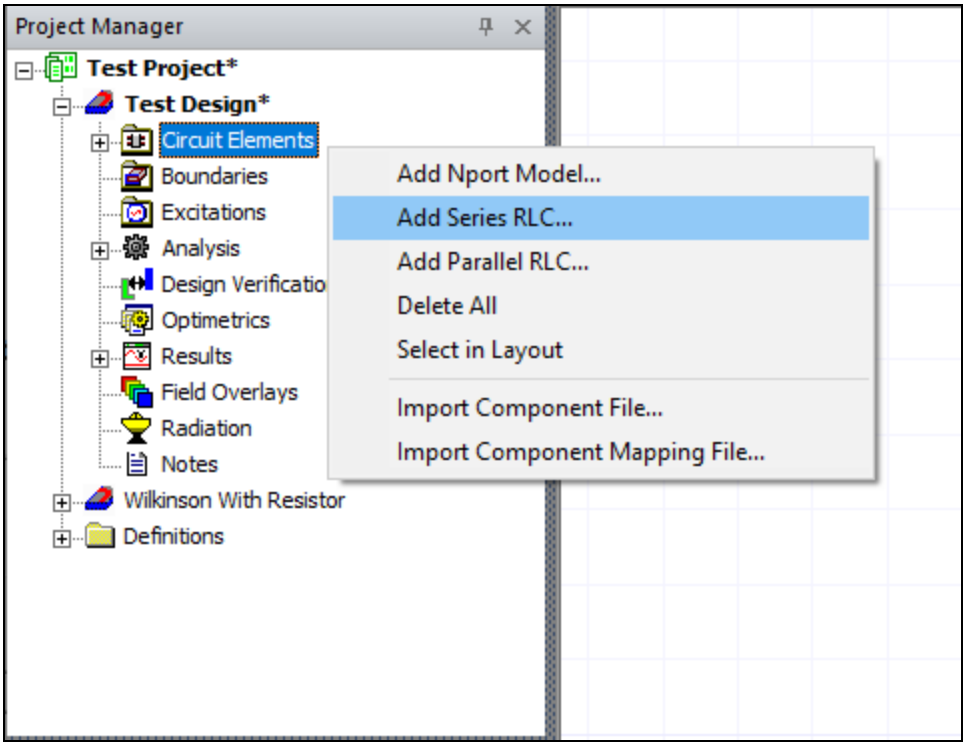
Typically, the intersection should be corrected before proceeding. Users can change the shape of objects, or remove a component to reach this goal.

Creating a Circuit Element

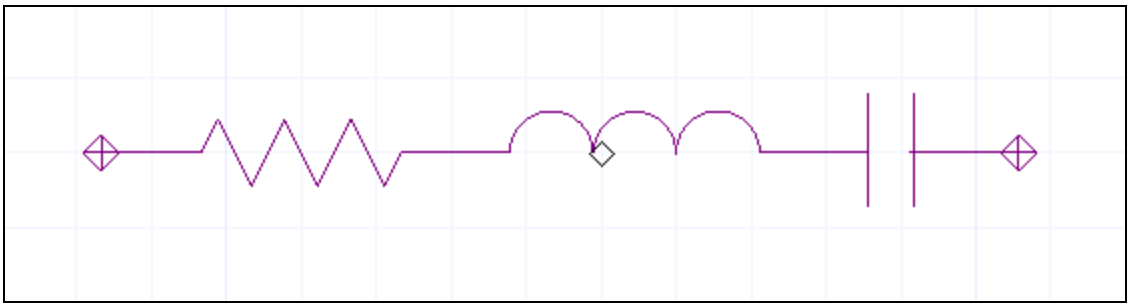
Follow these steps to add an RLC circuit element to a design.

1. From the **Project Manager** window, expand the **Project Tree** > **[Active Design Folder]**. Then right-click **Circuit Elements** and select **Add Series RLC** or **Add Parallel RLC**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

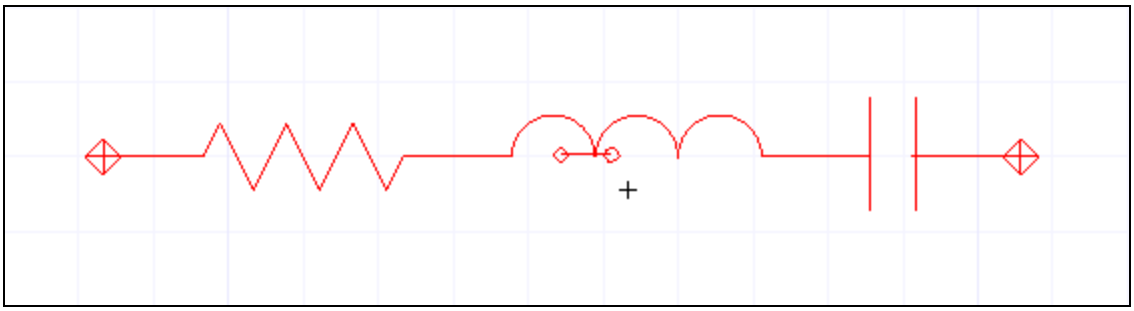


The selected element appears in the **Layout Editor** fixed to the cursor.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

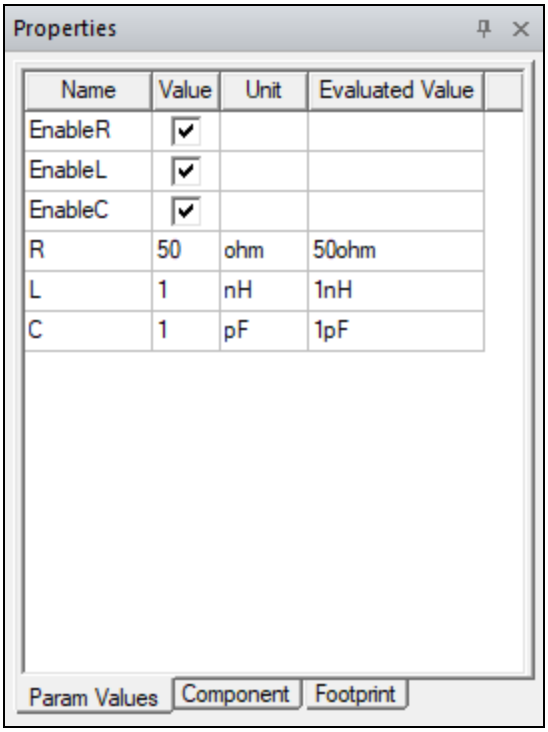
2. Click in a selected place in the design to place the element.



Modifying a Circuit Element's Properties

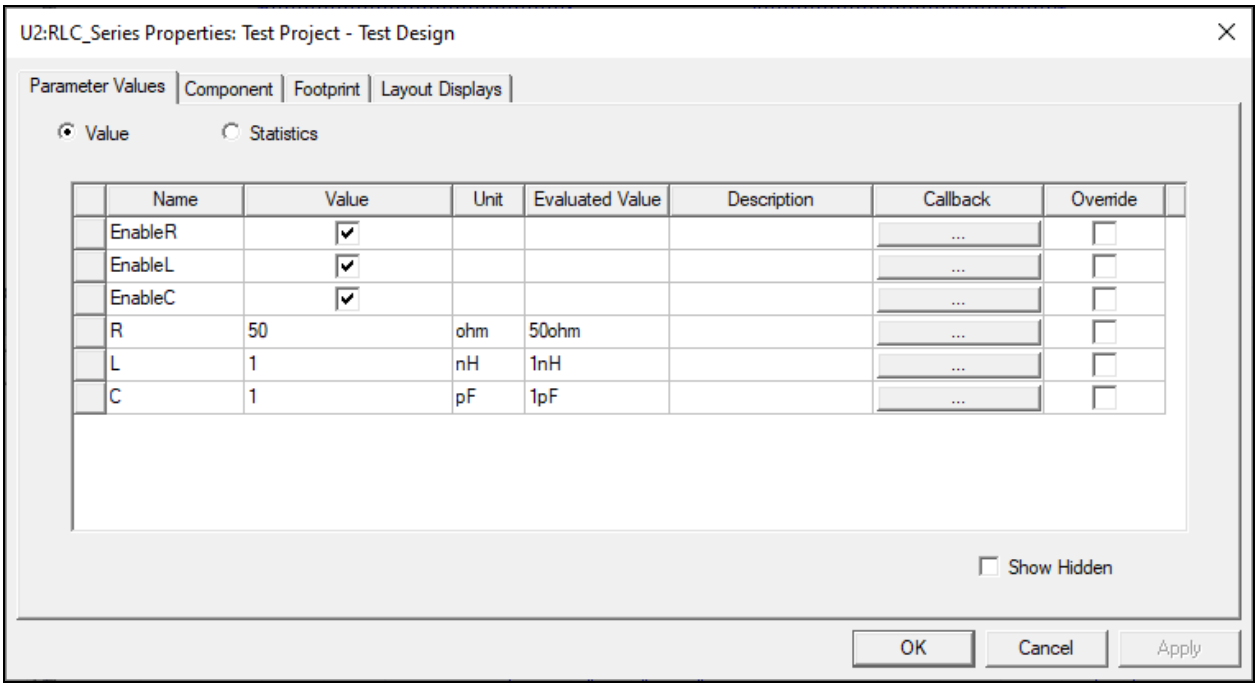
Modify a circuit element's properties by doing either of the following from the **Layout Editor**:

- Select the circuit element and change the selected 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.

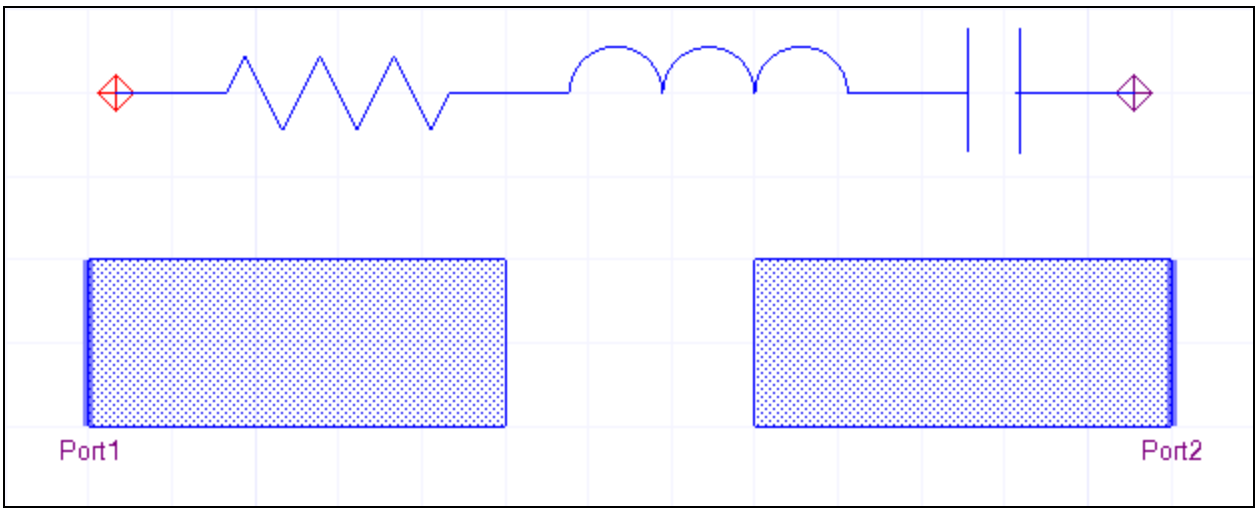
- Double-click the element to open the **Properties** window.



Connecting a Circuit Element

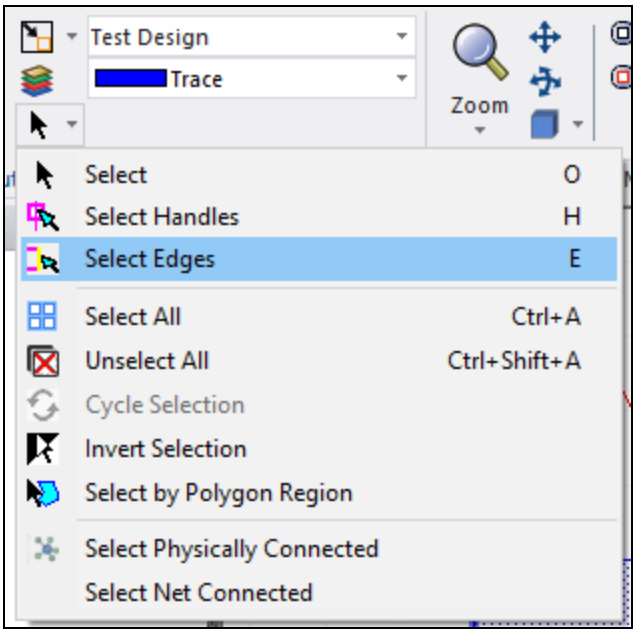
Follow these steps to connect a circuit element to an existing design element.

1. Select the outside edge of one of the circuit element's terminal ports. If the entire circuit element is selected, press B to switch between the selectable regions of the circuit element (i.e., the entire circuit element or only the terminal port).

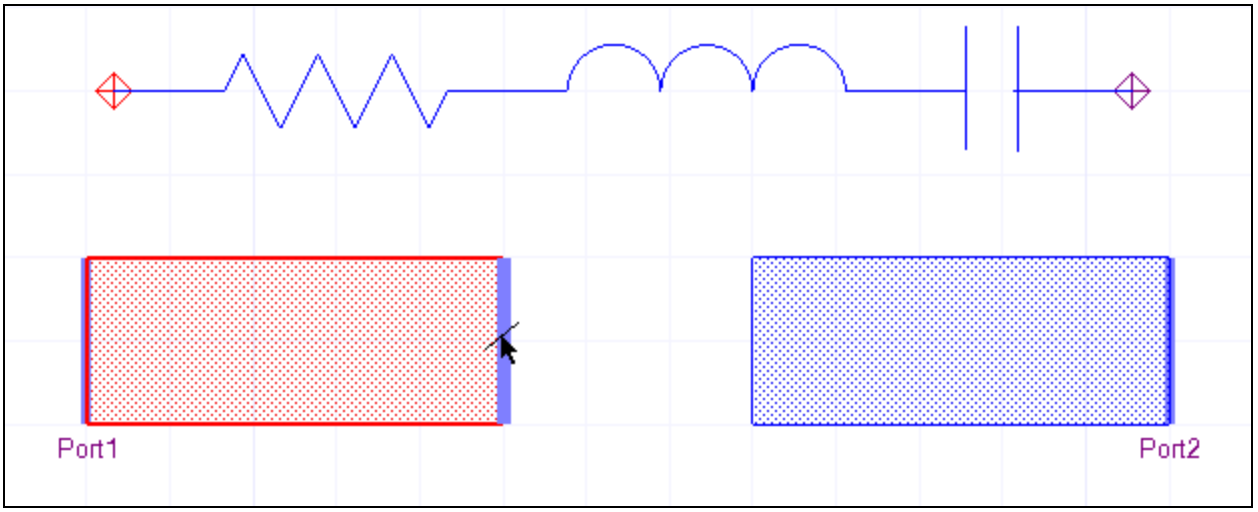


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. To enter **Select Edges** mode, either type **E** or navigate to the **Select elements** drop-down menu and click **Select Edges**.



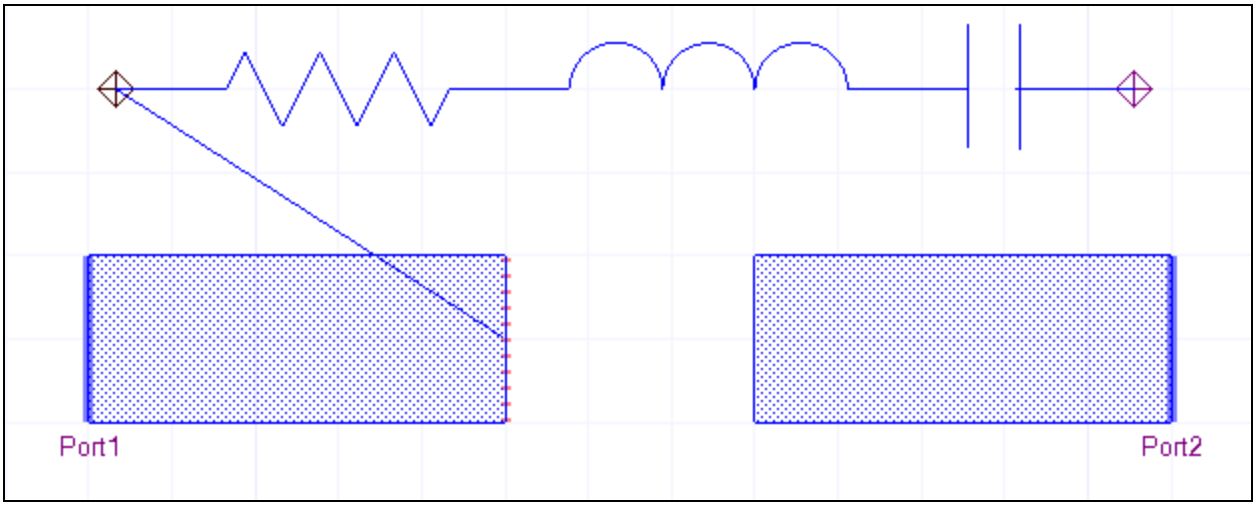
3. Click an unused edge of the design to ready it for port entry.



4. From the **Layout** ribbon, select **Connection**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The circuit element is immediately connected to the existing design element.



> **Note:**
>
> If the connection is not visible, ensure visibility is selected for **Rats** in the **Layout Editor**. Repeat the same procedure to define a connection for the second terminal.

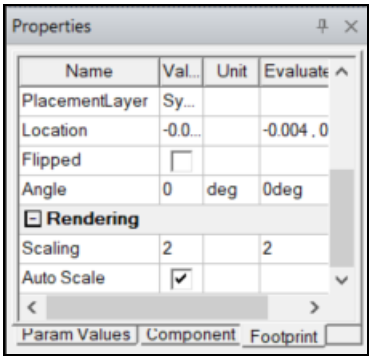
5. Repeat steps 1-4 to connect the circuit element's second terminal port to the existing design element.

n

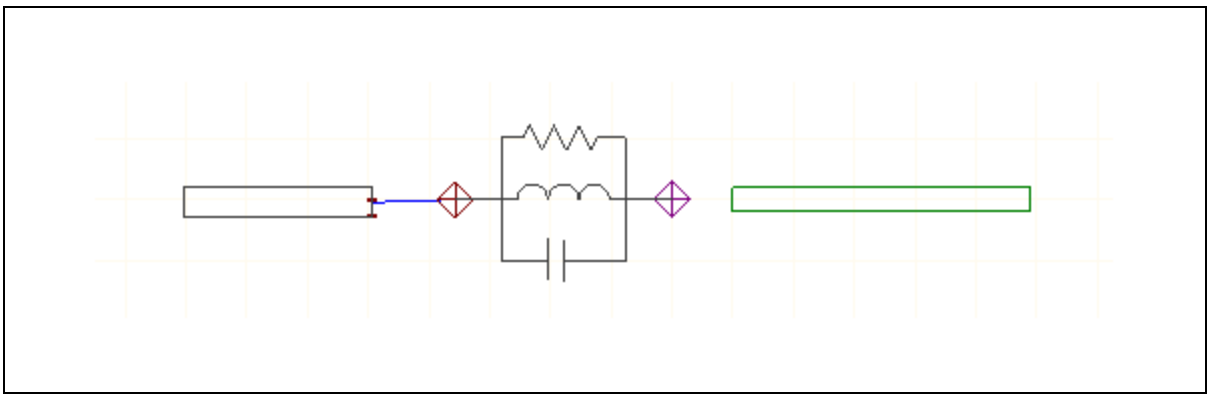
Auto Scale of Circuit Elements

When Auto Scale is enabled for a circuit element and two or more of the circuit element's pins are connected, the circuit element is automatically scaled to fit the smaller of the dimensions of the bounding box of the connections.

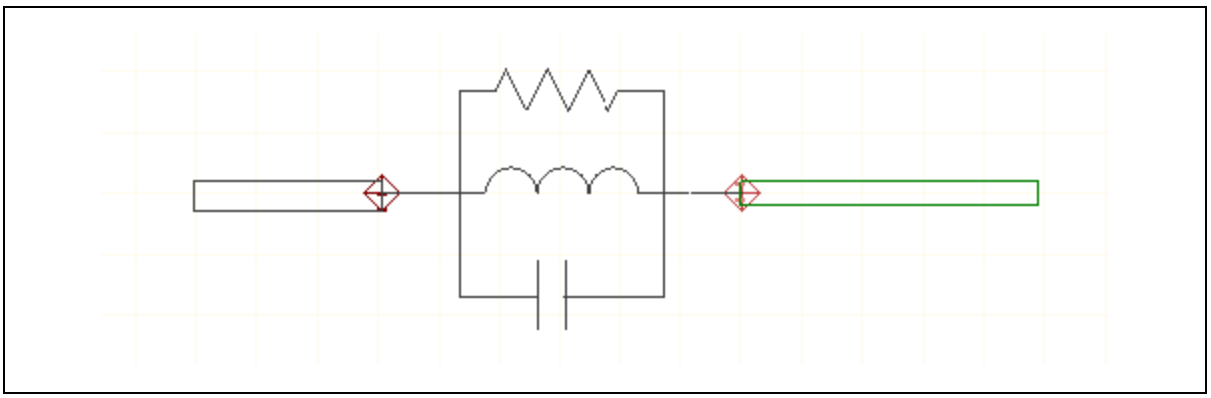
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure is a circuit element with one connection.



Auto Scale is enabled (checked), so when the second connection is made, the circuit element is scales to fit, as shown in the following figure.

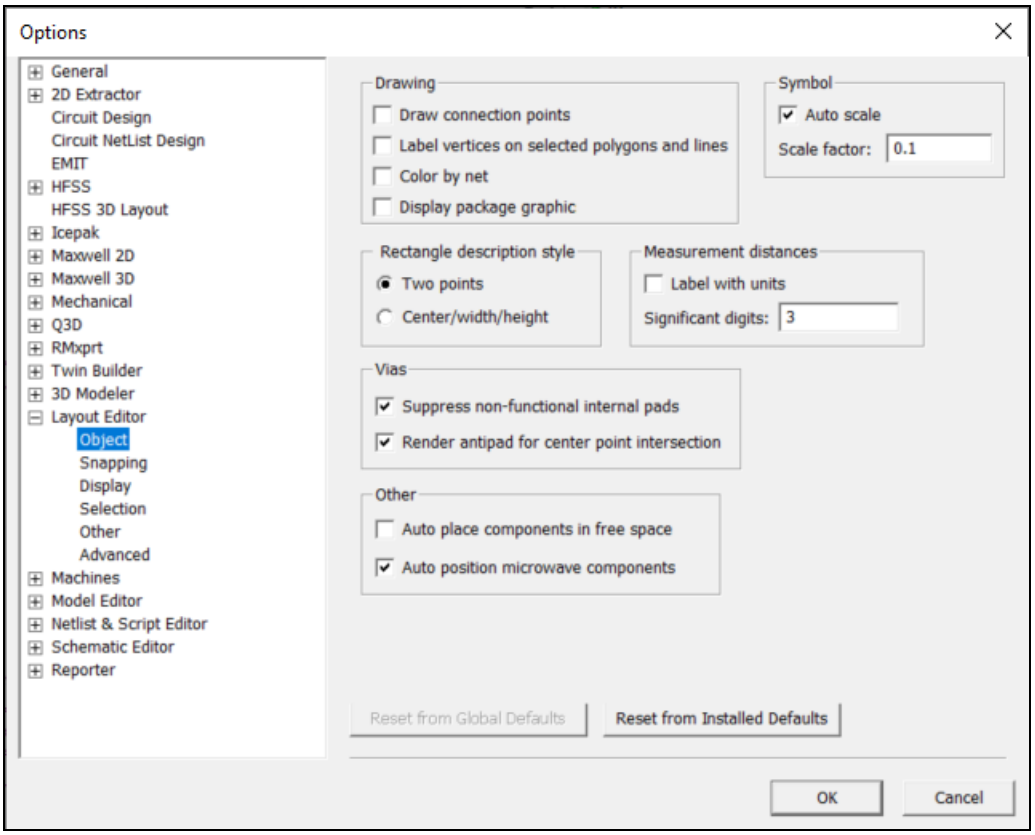


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turn Off Auto Scale

The circuit element does not move, so manually adjust the components as appropriate to match the pins to the connected location.

1. Auto scale is enabled by default. Follow these steps to deactivate Auto scale or adjust the scale factor (Scale factor is 0.1 by default).
2. Navigate to **Tools** > **Options** and select **General Options...** to open the **Options** window.
3. Expand **Layout Editor** and select **Object** to see the associated options in the **Symbol** area.
4. Click the **Auto scale** check box to remove the check mark and deactivate Auto scale. If applicable, type a new number in the **Scale factor** field to change the scale factor.

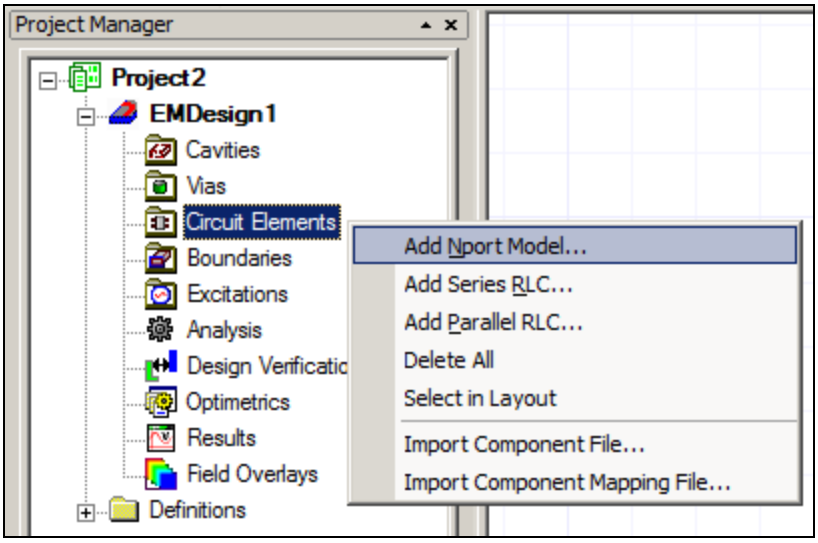


The process is complete.

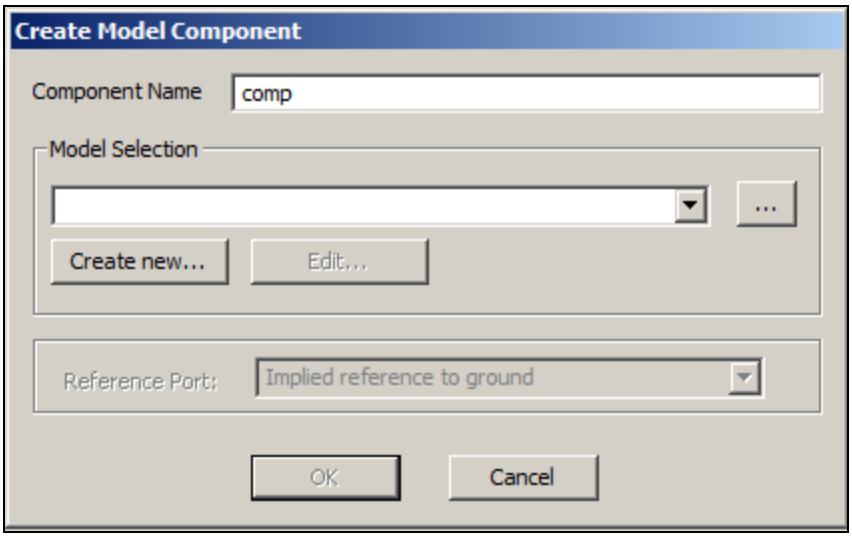
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nport Circuit Elements

To add an Nport model to a design, right-click **Circuit Elements** in the **Project Manager** window and select **Add Nport Model**.

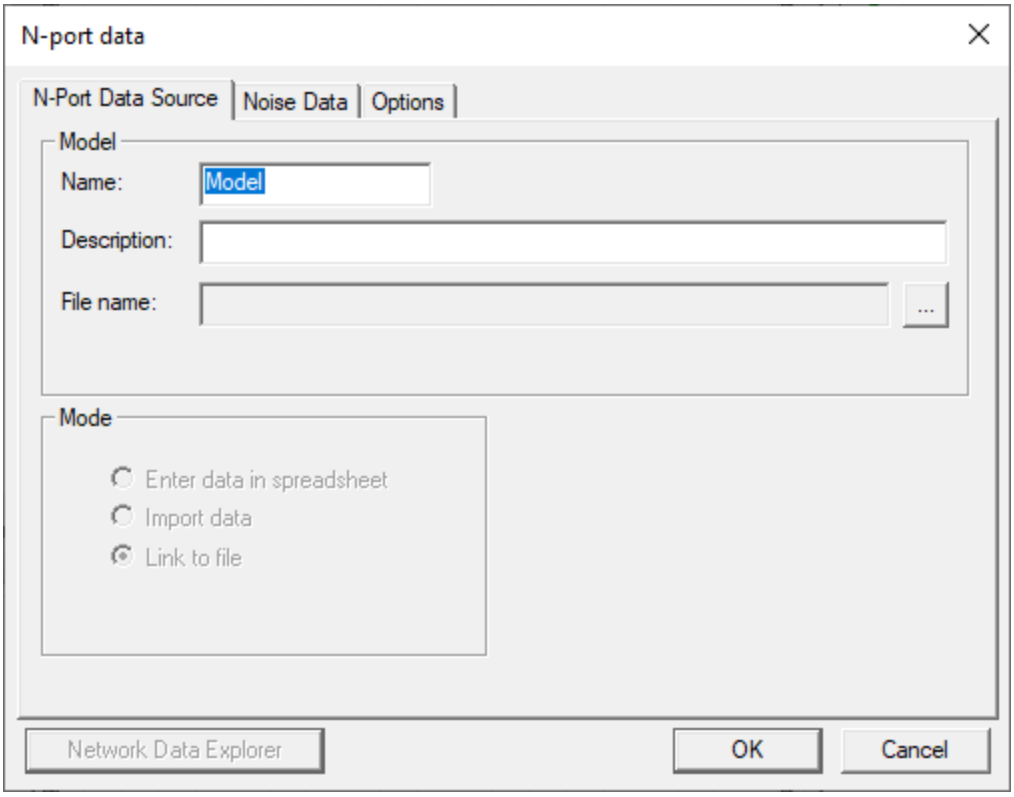


The **Create Model Component** window opens in which the component Model can be created or edited, and an additional option controls the reference used to determine port voltages.



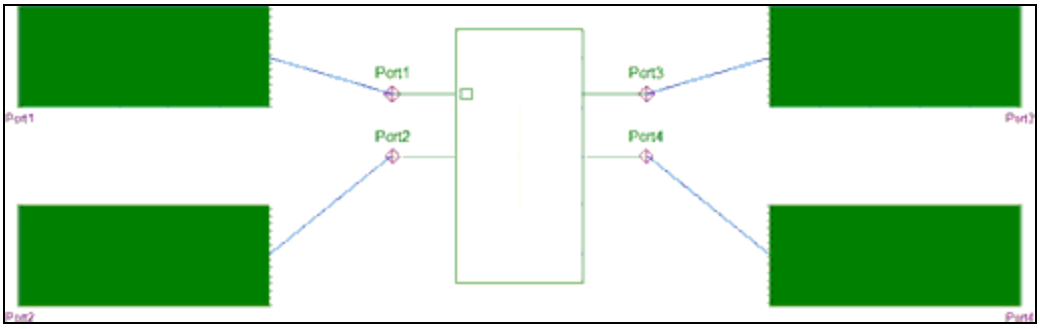
To create a new model, click **Create new** to open the **N-port data** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



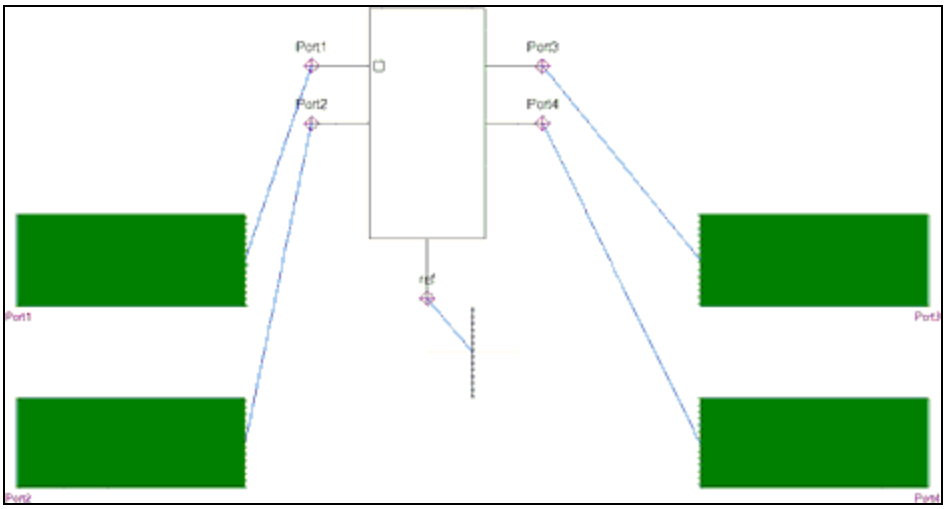
From this window, the component can either be defined by importing or linking to an existing definition in a touchstone file, or by manually entering the S, Y, or Z Matrix entries into a spreadsheet.

Once the model has been created the component can be placed into a design and the ports connected to edges of conductors as was the case for series and parallel RLC elements. There are three cases to consider, based on the type of Reference Port selected in the **Create Model Component** window. For **Implied reference to ground**, each of N port connections must be defined.

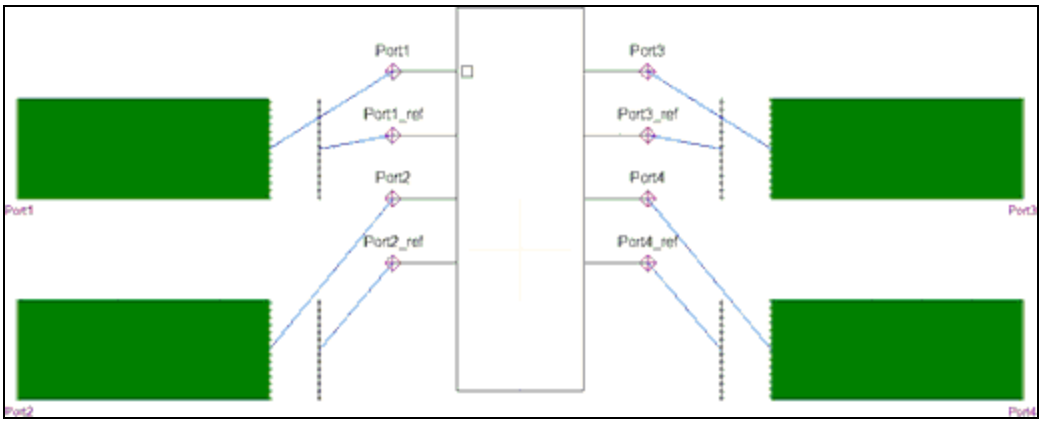


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **Show common reference port**, an additional connection must be defined representing a common reference for the definition of port voltages.



In this case the common reference is an edge in the ground plane. For **Add individual reference pin per port**, an additional N connections must be defined allowing for a different reference for each port.



In this case, each port is referenced to a nearby edge in the ground plane.

Creating a Circuit Element Between Any Two Layers

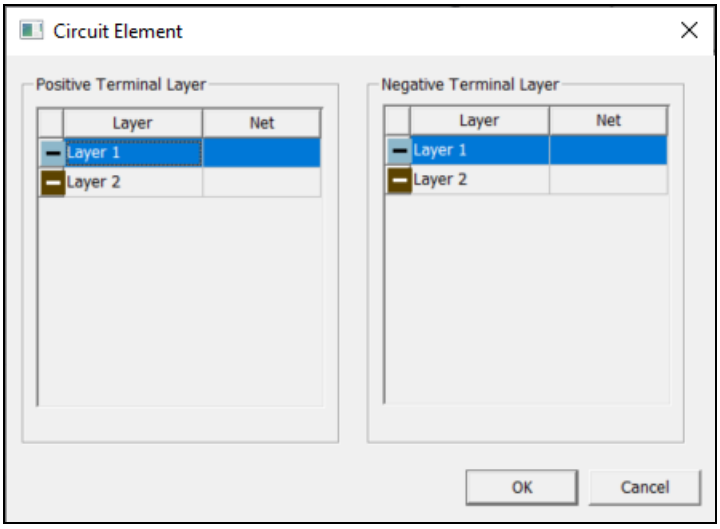
When most components are created, they are formed on a single layer from existing pins. Combinations of resistors, inductors, and capacitors can be created without pins and with terminals that are attached to different layers.

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 circuit element is created, a generic graphic appears. As the RLC properties are deactivated or deactivated, the graphic updates to reflect the state of the element. The element's model type determines if the graphic displays elements in series or in parallel. A circuit element which has an S-parameter model assigned to it is represented by a generic graphic with an X in it.

Follow these steps to create a circuit element between two layers. These steps assume that there are already multiple layers in the project.

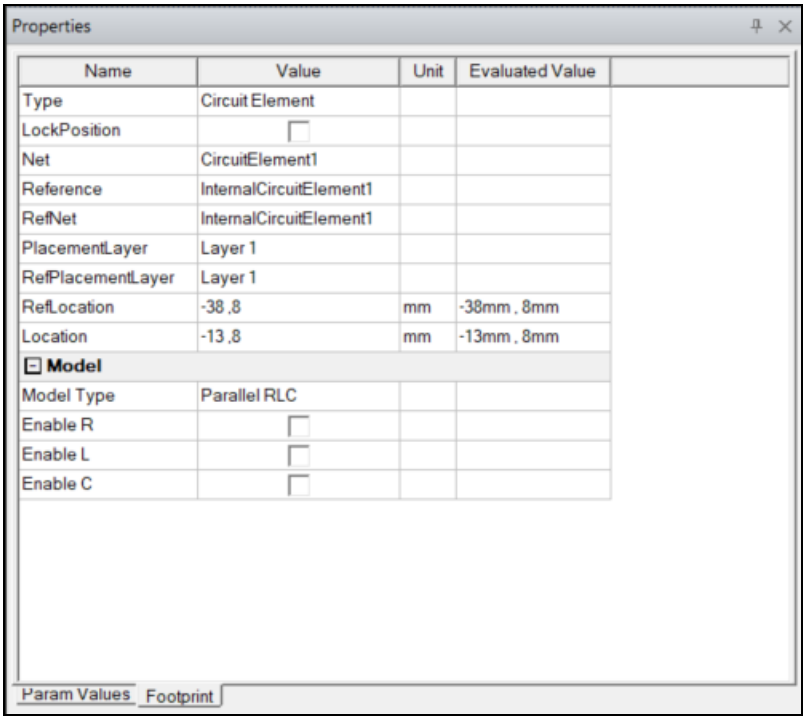
1. From the **Layout Editor**, right-click and select **Boundary** > **Create Circuit Element**.
2. **Click+drag** to draw the component into position in the **Layout Editor** to open the **Circuit Element** window.



3. Choose which layer each terminal of the component connects to and click **OK**.
4. Select 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. From the **Properties** window, click the **Footprint** tab.



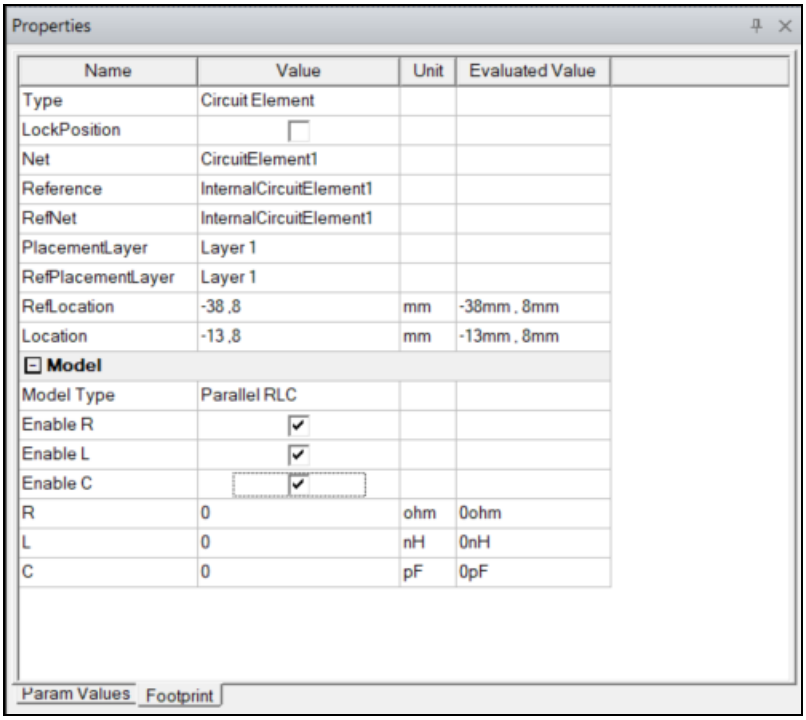
6. Choose one of the following model types on the **Model Type Value Field** drop-down menu:
    - **Parallel RLC** – the components are parallel to one another
    - **Series RLC** – the components are in series to one another
    - **S-parameter** – a frequency-dependent, imported model

Parallel RLC or **Series RLC** components only.

1. Add a resistance:
    a. Click the **Enable R** check box to add an **R** row to the **Footprint** tab.
    b. Click the **RValue Field** to enter the resistance into the field.
2. Add inductance:
    a. Click the **Enabled L** check box to add an **L** row to the **Footprint** tab.
    b. Enter the inductance into **L**.
    c. Click the **L Value Field** to enter the inductance into the field.

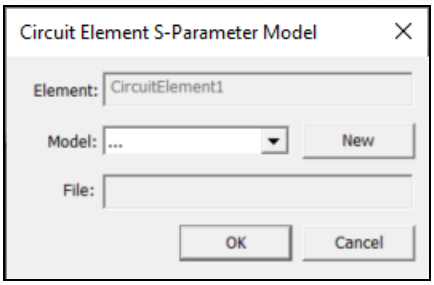
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Add capacitance:
    a. Click the **Enable C** check box to add a **C** row to the **Footprint** tab.
    b. Click the **C Value Field** to enter the capacitance into the field.



S-parameter components only.

1. Click the **S-parameter mo del [...]** button in the **Value** field to open the **Circuit Element S-Parameter Model** window.



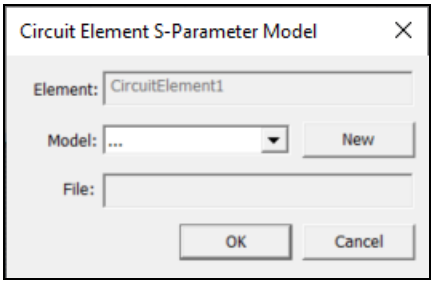
2. To import an existing model, click the **Model** drop-down menu and select an existing model. By default, no model is selected. Then click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Import a new model, click **New** to open an explorer window. Then navigate to the model, select it, and click **Open**.

> **Note:**
>
> By default, the model is assigned the same name as the file. To give the model a new name, enter it in the **Comp Name** field.

Click **Open**. The new model is imported into HFSS 3D Layout and set in the S-parameter model row.



3. The new model is imported into HFSS 3D Layout and set in the **S-parameter model** row.

Physical Definition for EM Models

To verify that a component possesses a physical definition, access the Edit Component window. This can be accomplished one of two ways, depending upon whether the component is user-defined or is part of the HFSS library.

User-Defined Components

If the component is contained in a loaded project:

1. From the **Project Tree**, expand the **Definitions** folder and the **Components** subfolder for the project that contains the component you want to examine.
2. Double-click the component entry, or right-click the component entry and select **Edit Component** to open the **Edit Component** window.

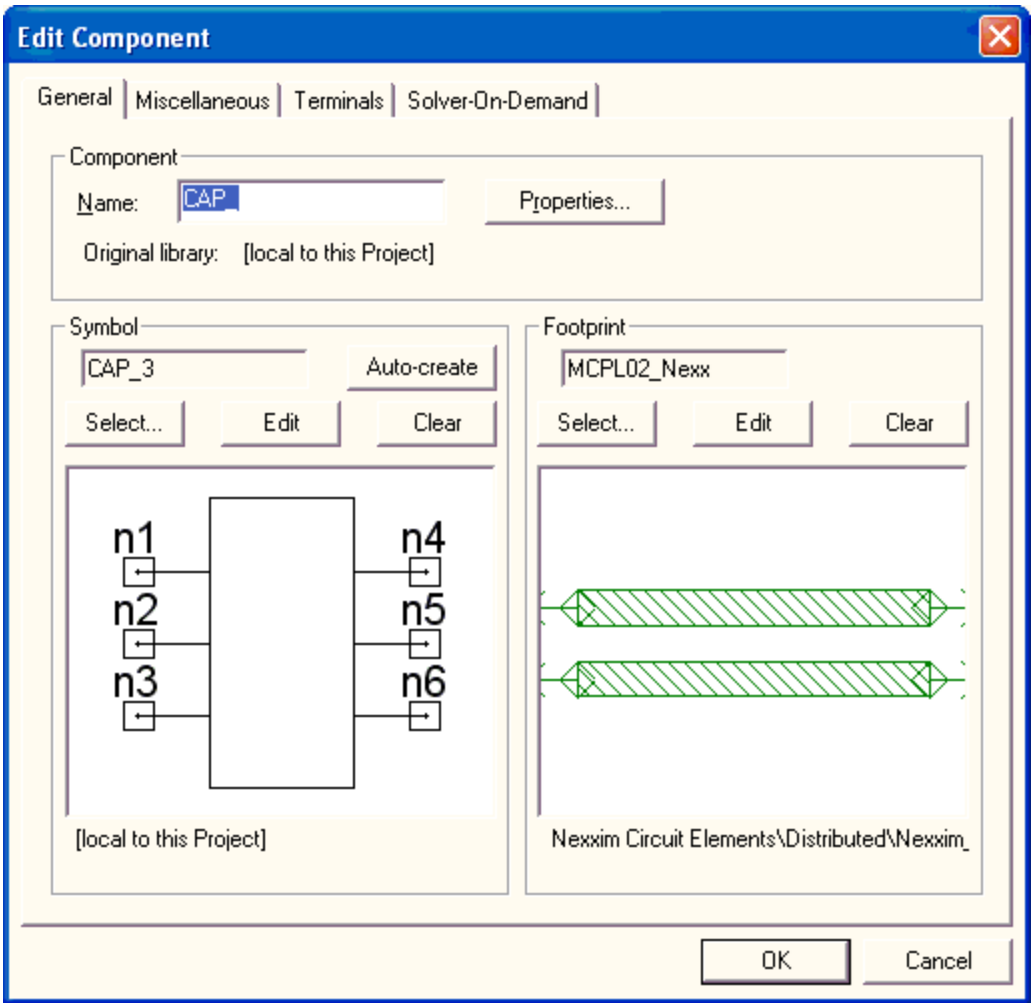
Library Components

If the component is contained in a library known to HFSS:

1. Select **Tools** > **Edit Libraries** > **Components** to open the Edit Libraries window, then select the component you want to edit.

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 Component**, or double-click the selected entry, to open the **Edit Component** window.



The physical definition of a component can be verified using the information displayed by the window. In particular, information related to the footprint of a component is displayed.

Working With The Capacitor/Inductor Library Browser

Ansys SIwave™ allows you to define VRM parameters and impedance requirements, then automatically select capacitors that meet those requirements. Also plot the impedance of various vendor or imported capacitor components versus frequency.

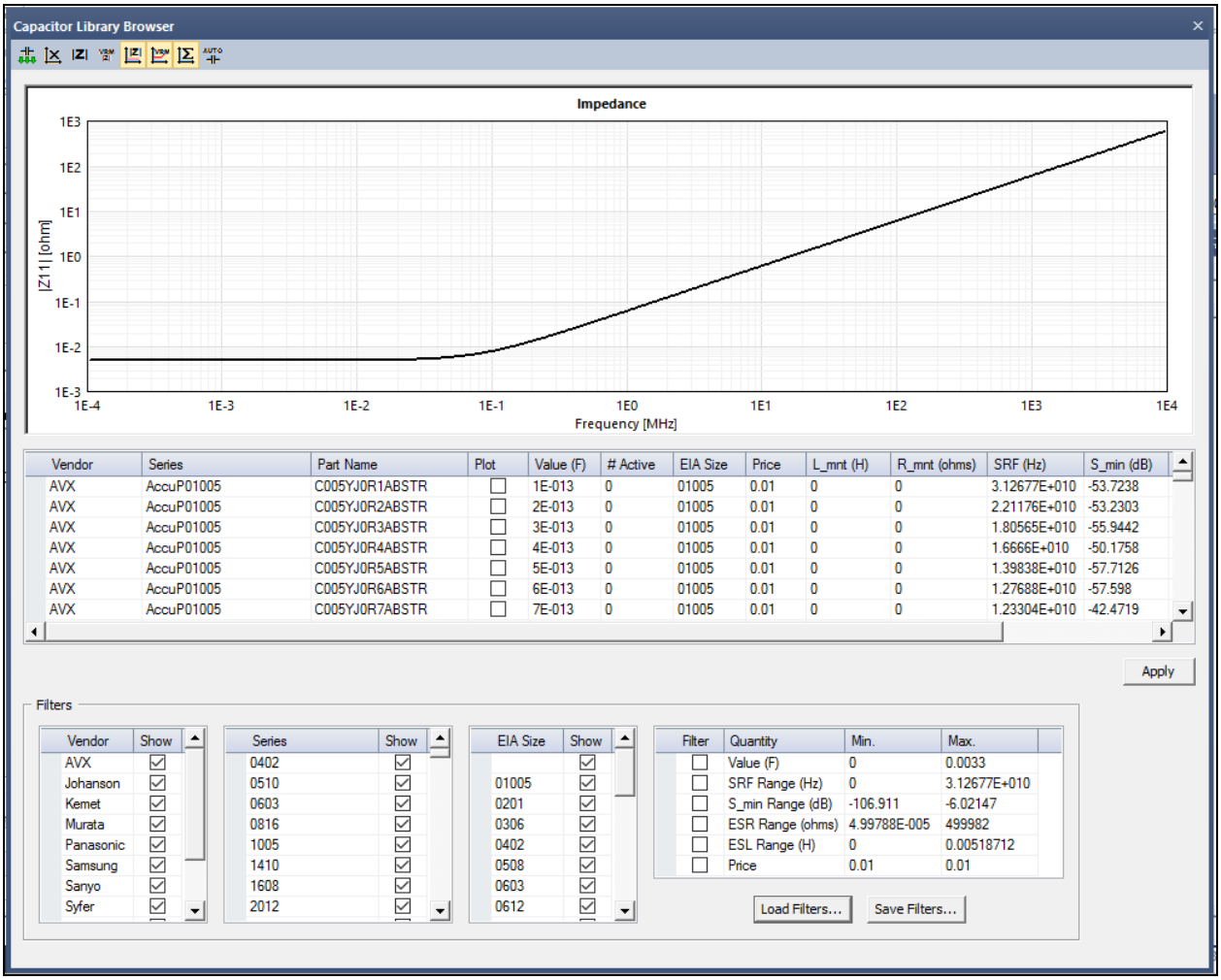
To view the Capacitor/Inductor Library:

1. Select a capacitor in the **Components** window, click **Model Info** for the capacitor in the **Properties** window, then in the **Component Model** window that appears, click the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

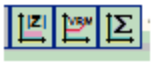
Library Browser to open the **Capacitor/Inductor Library Browser** window.

You can sort the values in each column by clicking the column title.



2. To automatically select the capacitors that match the defined impedance profile, click the icon.
3. To manually select the capacitors, click the **Plot** check box. To plot all the capacitors, click the **Plot** column title.
4. Specify filters for the options that you want to show. You can save these options to a **.cff** (capacitor filter criteria) file. Also load an existing filter.
5. To update the value for multiple capacitors, make a selection on the value drop-down menu. Then enter a value for all the selected capacitors in the field, then click **Update**.

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 import a capacitor model from a Touchstone file: click the icon to open the **Import S-parameter Component** window. Specify the component details, and click **OK**.
7. To remove all capacitor impedance curves that have been selected, click the icon.
8. To define the impedance requirements, click the icon to open the **Define Impedance Mask** window. You can create and save the existing profile, or load a saved profile.
9. To define VRM parameters, click the icon to open the **Define VRM Parameters** window. Specify the ESL and ESR parameters, along with the plot color, and click **OK**.
10. To change the display options, click the corresponding  icons to display impedance mask, VRM output impedance curve, or aggregate output impedance curve.
11. To update the value for multiple capacitors, make a selection on the value drop-down menu. Then enter a value for all the selected capacitors in the field, then click **Update**.

Add a New Member to a Component

When entities are added to an existing component, the component's border shifts accordingly and its definition changes if new pins are added. From the **Layout Editor**, right-click the component and entity that you want to add to it and select **Components** > **Add Pin(s)**.

> **Note:**
>
> If a pin is added to a component with an RLC network model type, the pin pairs cannot be updated.

Remove a Member from a Component

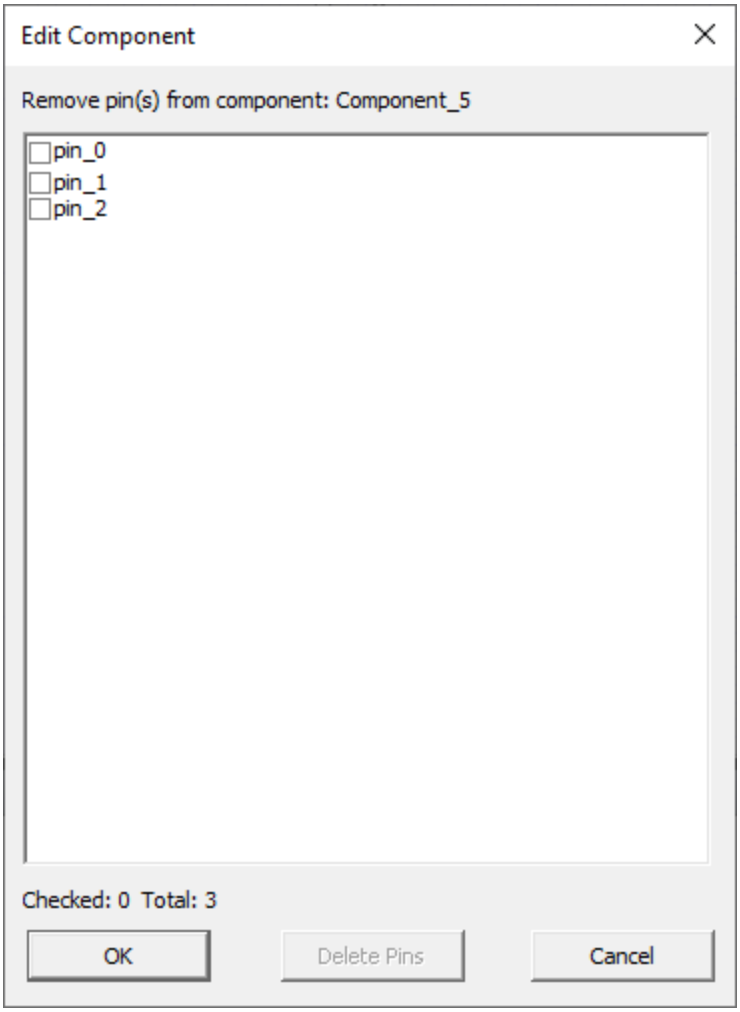
You can remove entities from components. When you do, the component's border shifts accordingly and its definition changes if pins are removed. Depending on the component's model type, you may need to make other changes when removing a member of the component.

To remove a single member: in the **Layout Editor**, right-click the unwanted member in the component and in the shortcut menu, select **Components** > **Remove Pin(s)**.

To remove multiple members:

1. Highlight the component.
2. From the **Properties** window, select **Edit** in the **Pins** row to open the **Edit 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.



3. From the **Edit Component** window:
    a. Select the unwanted pin(s).
    b. Click **Delete Pins**.
    c. Click **OK**.
4. Check the component's model type for whether any updates

If the component is redefined and has a SPICE model that requires more pins, the redefined component has, then update the model, reassign the pins, or give the pins another location.

When a pin is removed from a component with an RLC network models, its pin pair is removed on the **Model Info** pane in the Component Model window.

When a pin is removed from a component with a S-parameter model, the exact model is removed on the component. A new S-parameter model must be assigned 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.

Dissolve a Component

Delete a component by dissolving it in one of two ways.

- From the **Layout Editor**, right-click the unwanted component and select **Components** > **Dissolve**.
- Highlight the component and in the menu bar, click **Draw** > **Components** > **Dissolve**.

The component's bounding box is removed on the **Layout Editor** and the component is removed on the **Component** window.

Importing and Exporting Layout Components

All entities that make up an active design can be packaged, exported, imported, edited, and shared as a single layout component, either encrypted or unencrypted. This includes entire layout components as well as individual parts (e.g., such as pins, vias, or primitives) on a layer that forms a chip or RLC. Once a layout component is created, it can be exported and shared with one of two encryption levels. If multiple layout components are selected for export as part of a layout component group, users can choose to expose specific layout components from the group while maintaining full encryption for the overall design. These exposed layout components will remain visible upon import, enhancing usability and interoperability while preserving encryption capabilities.

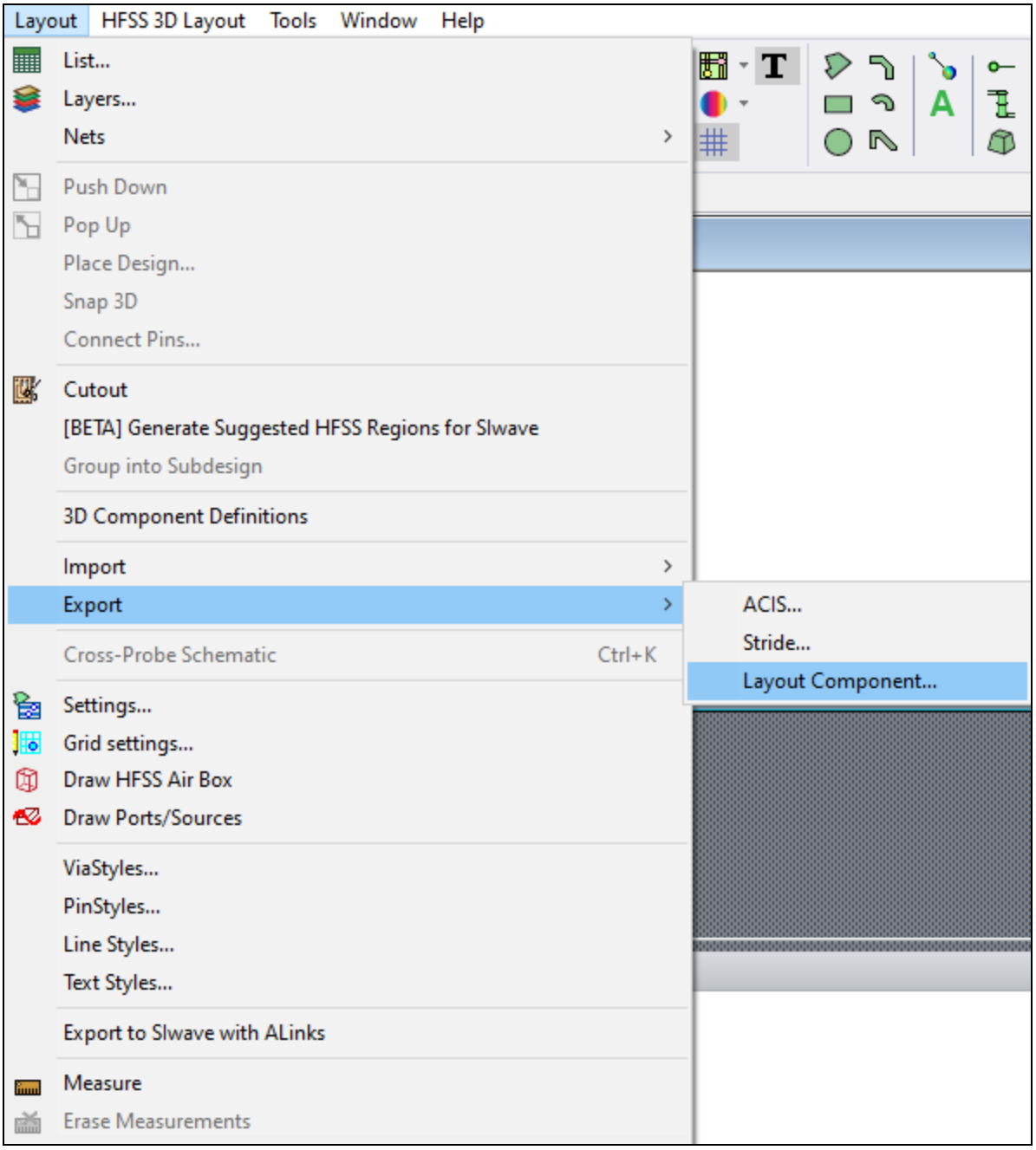
> **Note:**
>
> Field overlays are not supported and toggling layer visibility is unavailable for layers within exposed layout component groups.

Export and, if applicable, encrypt a layout component by completing these steps.

1. From the **Layout Editor**, select the layout component.
2. From the **Layout** menu, select **Export** > **Layout Component** to open an explorer

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

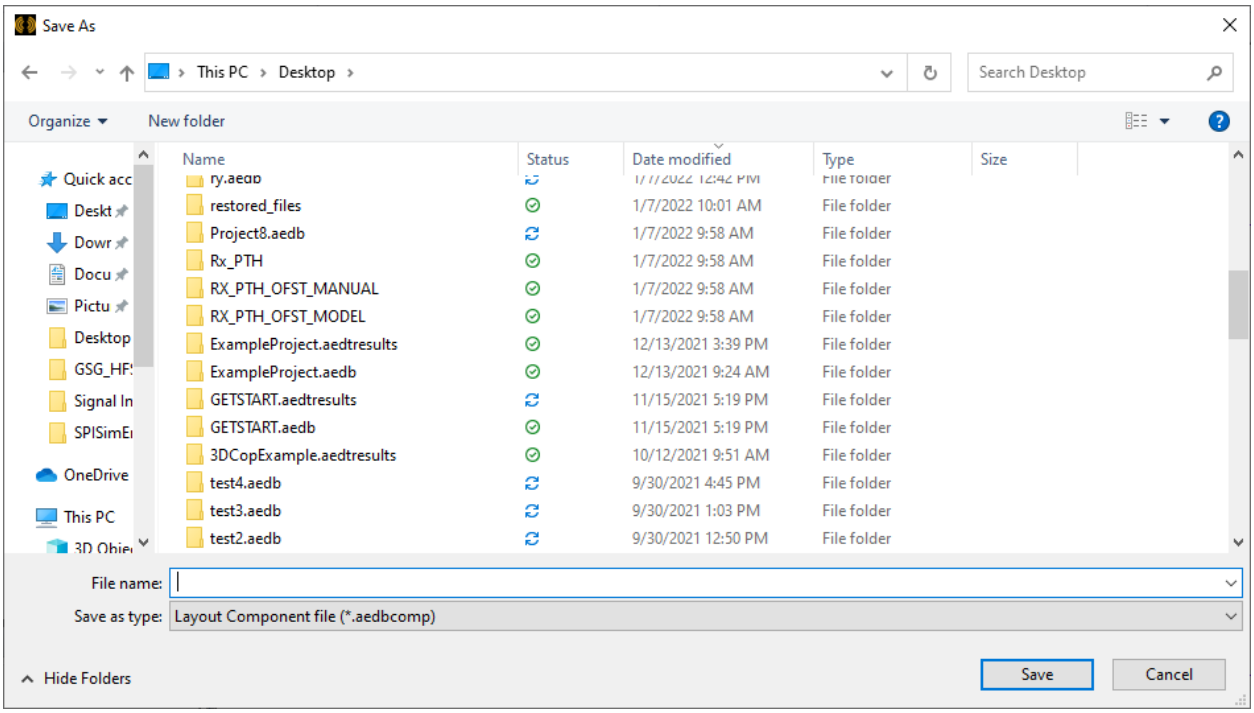
window.



3. Navigate to a selected directory. Then enter a **File name** for the layout component and, if applicable, choose **Layout Component file (*.*aedbcomp*)** from the **Save as type** drop-

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

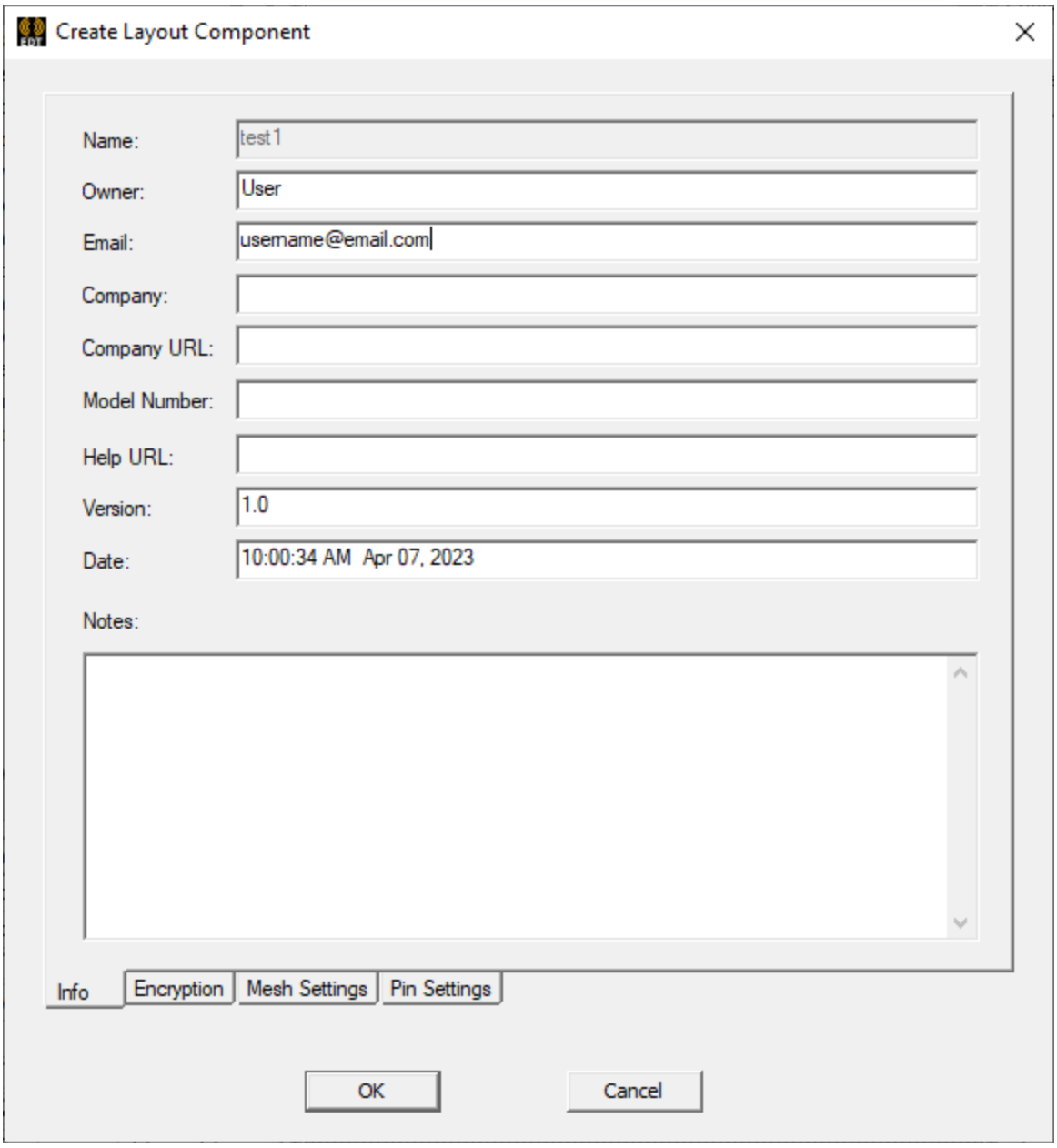
down menu (selected by default).



4. Click **Save** and the layout component will be saved in the selected directory in a folder named **<filename>*.aedbcomp***.
5. The explorer window closes and the **Create Layout Component** window opens. From the **Info** tab, enter metadata into the appropriate fields. The layout component's file name

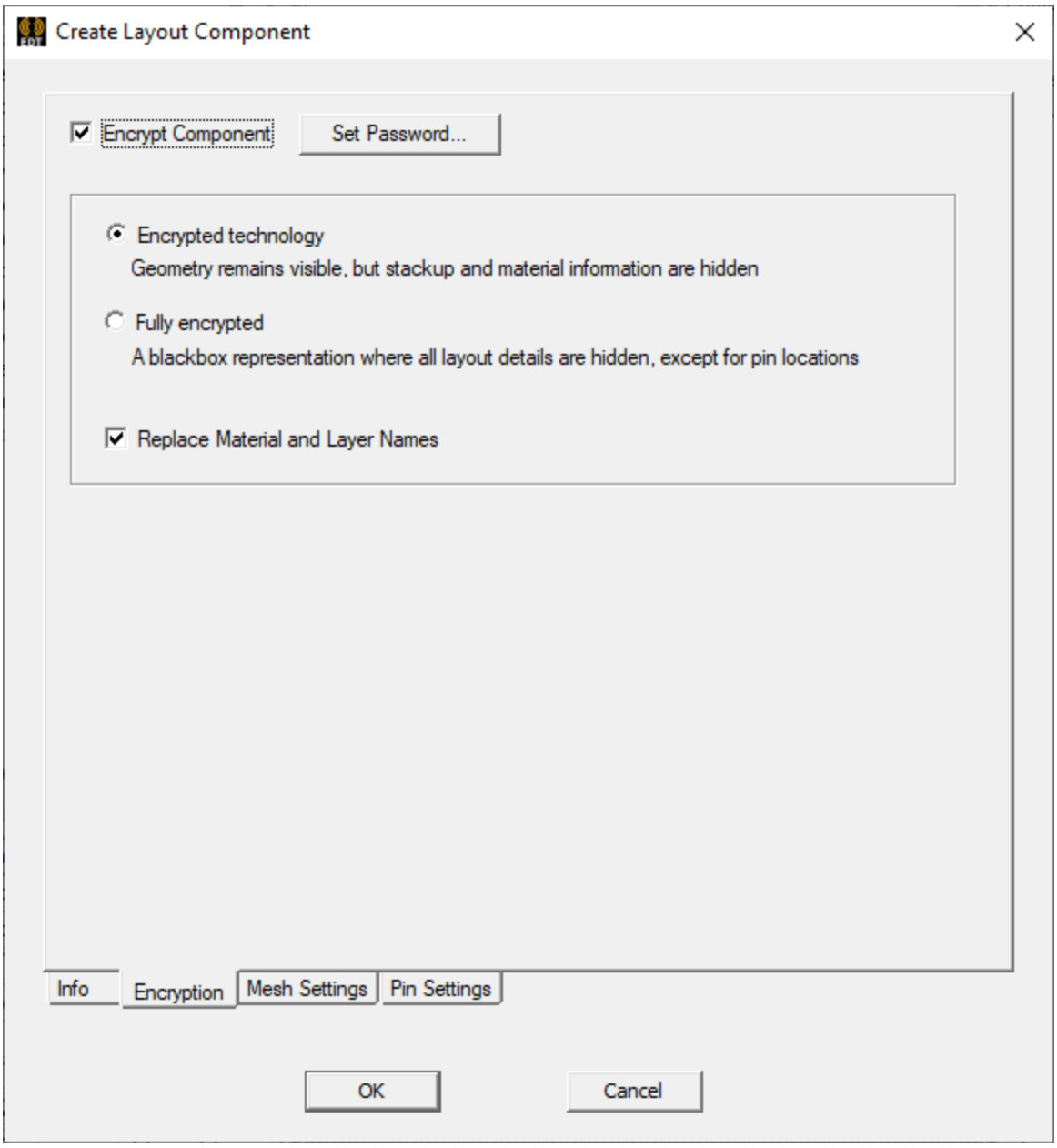
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is shown in the **Name** field.



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 the **Encryption** tab.

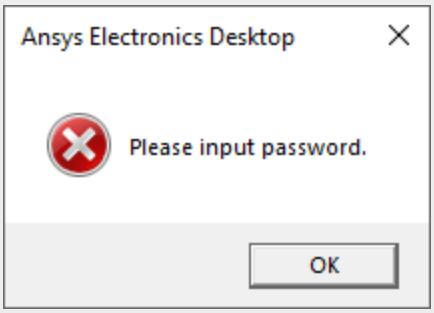


7. If no encryption is necessary, remove the check from the **Encrypt Component** box and skip to **step 13**. Otherwise, choose from the following:

   - **Encrypted technology** — geometry will remain visible, but stackup and material information is encrypted. The layout component may be editable. Click **Set Password** to open the **Set Passwords** window, and proceed to **step 8**.

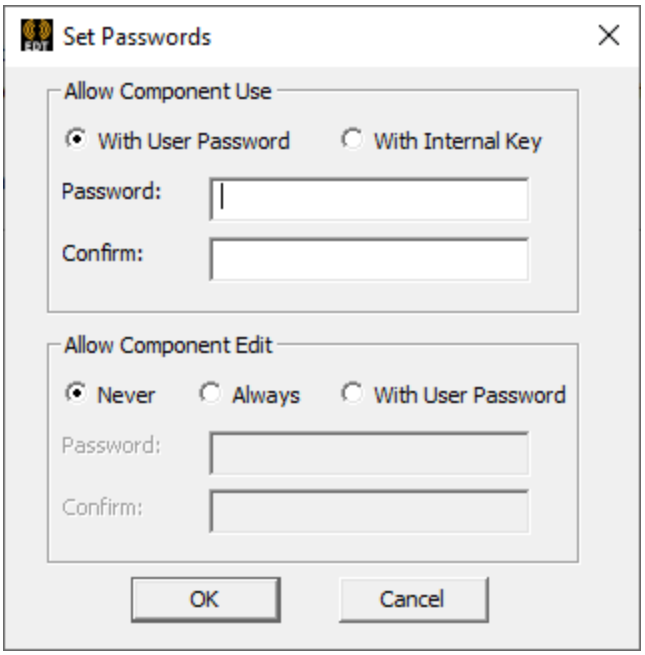
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Fully encrypted** — all data, except port locations, is hidden under a blackbox stencil. The layout component will remain uneditable. Click **Set Password** to open the **Set Passwords** window, and proceed to **step 8**.
- Check the **Replace Material and Layer Names** box and all material and layer names will be obfuscated (e.g., **Dialetric_1** will be renamed **Layer1** and the material column will be hidden from view).

> **Note:**
>
> If there is a check in the **Encrypt Component** box, the **Set Passwords** window will open if users attempt to leave the **Encryption** tab. Attempting to bypass the window will produce a **Warning: Please input password.** dialog box. This is true even if no encryption is selected. To leave the **Encryption** tab, uncheck the **Encrypt Component** box.
>
> 
>

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 open the **Set Passwords** window.



9. From the **Allow Component Use** area, choose from the following:

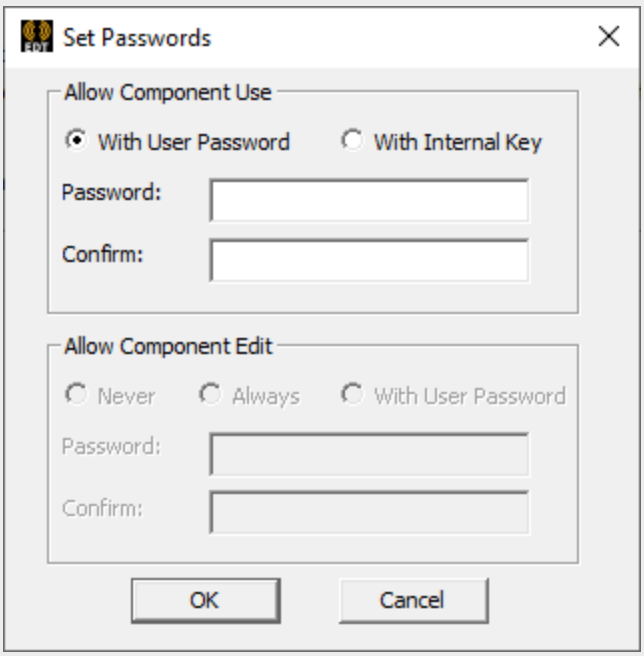
- **With User Password** — the layout component can be exported as an *.aedbcomp* file and reopened in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform or other compatible software with the correct password.
- **With Internal Key** — the layout component can be exported as an *.aedbcomp* file and reopened in (and only in) the Ansys Electronics Desktop (AEDT)™ electronics systems design platform without a password.

10. Enter and re-enter a password in the selected fields in the **Allow Component Use** area.
11. If future layout component editing is required, choose from the following in the **Allow Component Edit** area:
    - **Never** — once the **Set Passwords** window is closed, the layout component will no longer be editable.
    - **Always** — if future layout component editing is selected, and no further security is appropriate.
    - **With User Password** — if future layout component editing is selected, but some further security is appropriate. Enter and re-enter a password in the selected fields.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Choosing **Fully encrypted** deactivates the **Allow Component Edit** area. This level of encryption is a permanent state and the layout component will not be editable after the **Create Layout Component** window is finally closed.



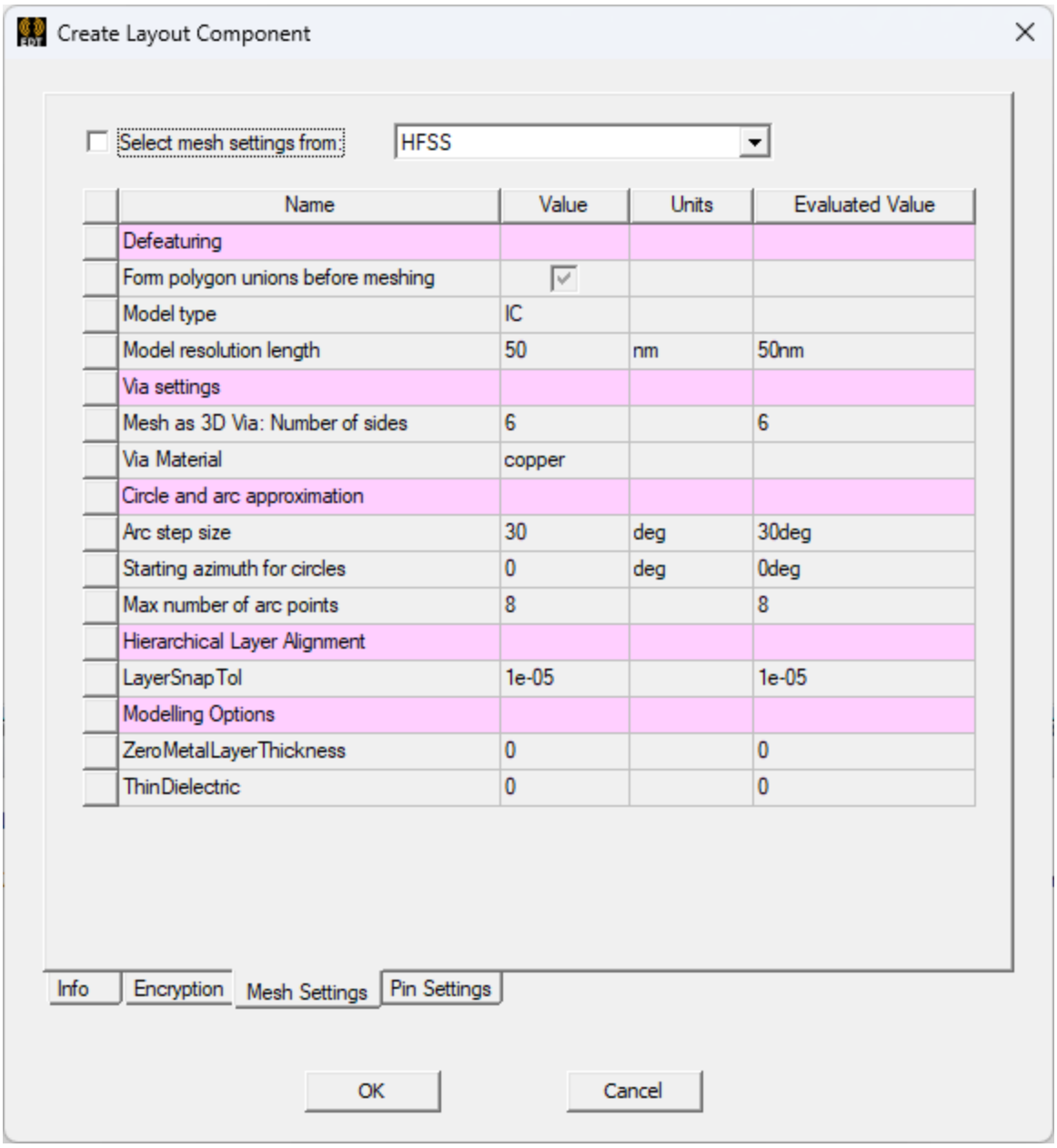
12. Click **OK** to save, close the **Set Passwords** window, and return to the **Create Layout Component** window.

Note:

Once a layout component is encrypted and users click **OK**, encryption cannot be removed from the layout component. However, encryption strength can be increased (i.e., if the user chooses not to encrypt the layout component, the exported layout component can be re-imported and exported again as an **Encrypted technology** layout component. If the user exports the **Encrypted technology** layout component, the exported layout component can be re-imported and exported again as a **Fully encrypted** 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.

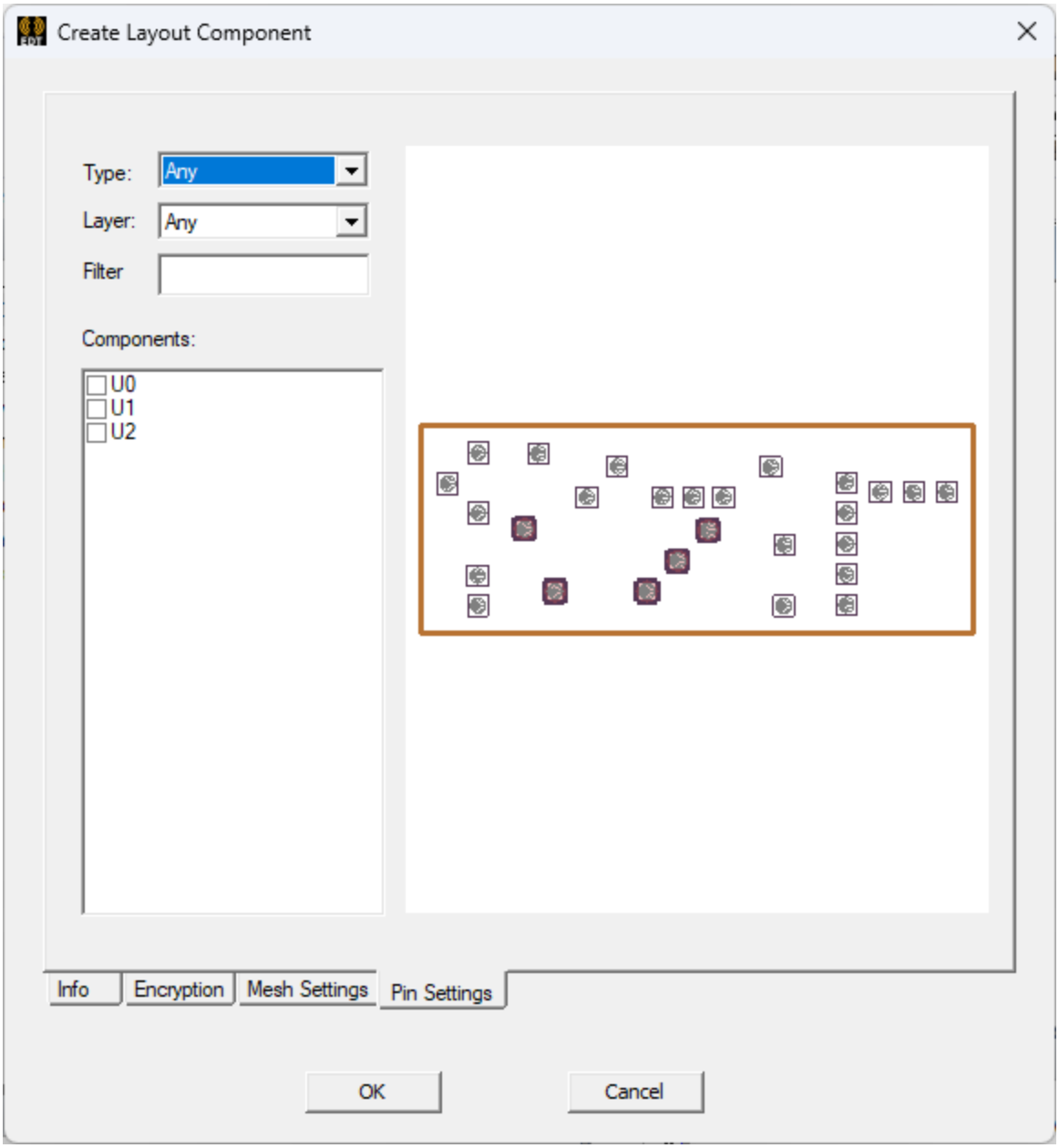
13. Click the **Mesh Settings** tab.



14. To save existing mesh settings with the exported layout component, check the **Select mesh settings from** box and make a selection from the drop-down menu.

15. If **Fully encrypted** is selected on the **Encryption** tab, click the **Pin Settings** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



16. From the **Pin Settings** tab, choose from the following:

    - Use the **Type** and **Layer** drop-down menus, as well as manually entering **Filter** variables to filter the list of available **Layout Components** selected on the **Layout Editor**.
    - **Components** — check boxes adjacent to each layout component part to leave the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

layout component(s) exposed, despite full encryption. The exposed layout components are shown in preview window.

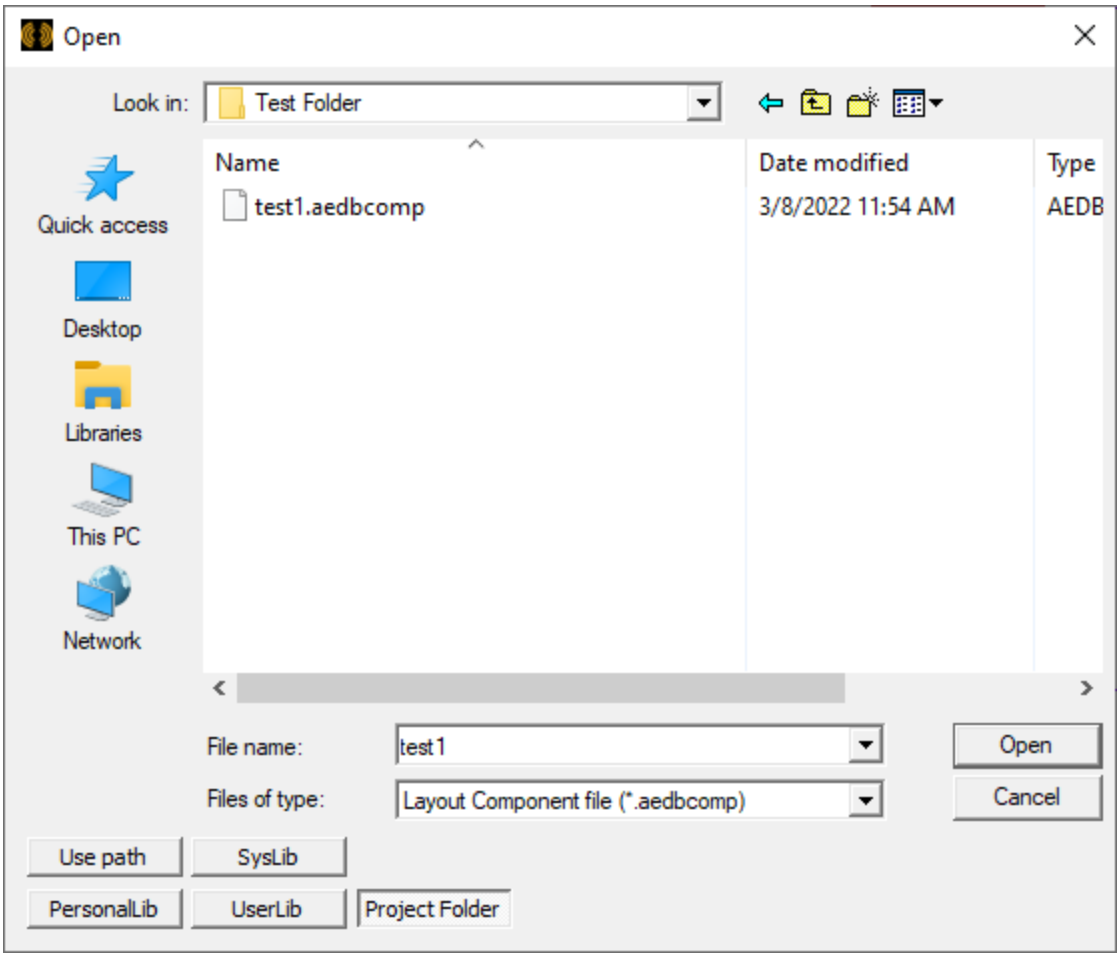
17. Click **OK** to export the layout component.

The procedure is complete.

Importing a Layout Component (General Mode Only)

Follow these steps to import a layout component into the active design.

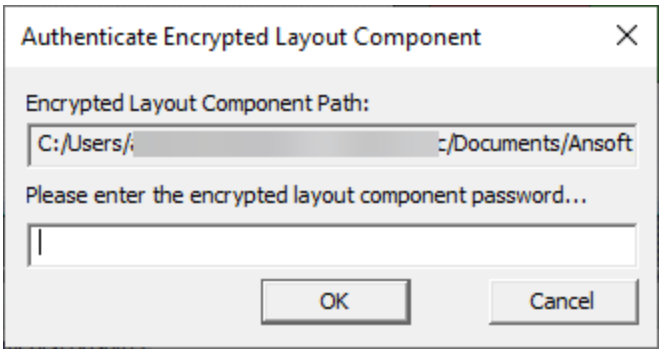
1. From the **Draw** menu, select **Import** > **Layout Component** to open an explorer window.
2. Navigate to a folder that contains the appropriate **Layout Component file (*.aedbcomp file)**. Then click **Open** to close the explorer window and return to the **Layout Editor**.



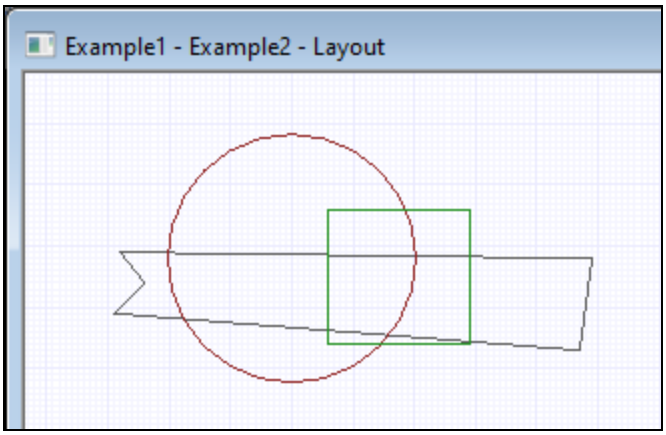
3. If the layout component is encrypted, an **Authenticate Encrypted Layout Component** window appears. Enter the layout component's encryption password, then click **OK** to

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

proceed.

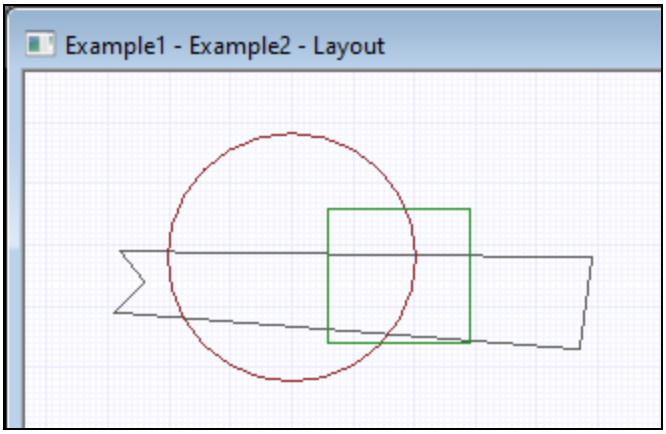


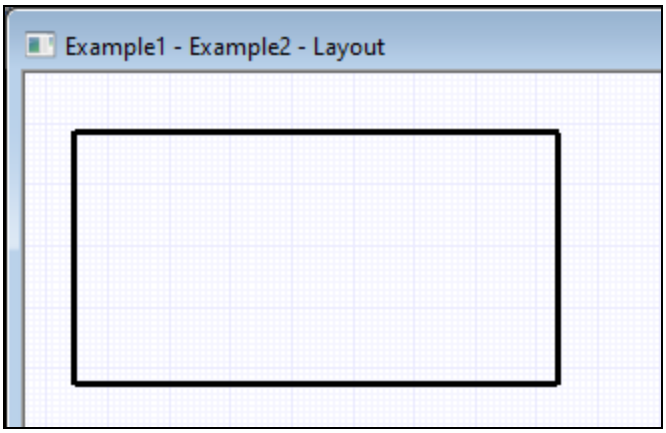
4. The imported layout component is attached to the mouse cursor as a stencil. **Click+drag** the stencil to move the layout component. Individual members of the layout component cannot be moved individually. Once a layout component is created, the positions of its members are frozen in relation to one another. If the layout component is unencrypted or partially encrypted, the stencil is an accurate representation of the layout component's geometry.



If the layout component is fully encrypted, the outline is a rectangle with no visible perimeters. Drag the stencil into position in the **Layout Editor** and left-click to place.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





The imported layout component is now part of the active design. A definition for it is created in the **Project Manager** window > **Project Tree** > **Definitions** > **Components** folder.

Visible Restrictions of an Encrypted Layout Component

An encrypted layout component has other obvious restrictions that users will notice at a glance, such as in the following examples.

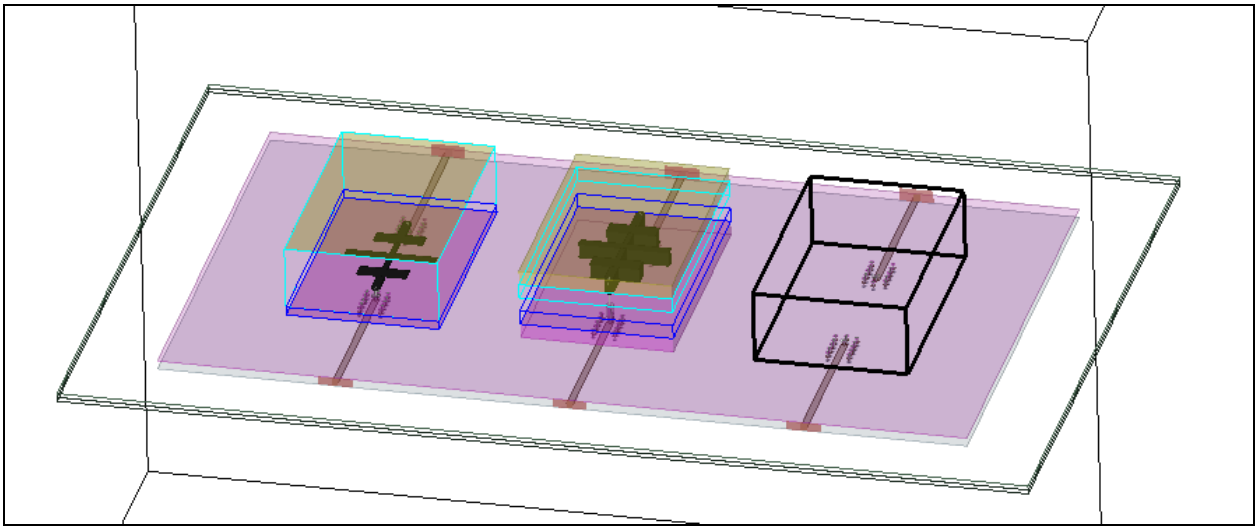
Layout Editor

Navigate to the **Layout Editor**. Layout Component stencils in the **Layout Editor** will differ greatly, depending on their level of encryption. Referring to the following example:

- **Unencrypted Layout Component** — (left-most) represents the design accurately, displaying every member of the layout component according to its material type, position, and dimensions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Encrypted Technology Layout Component** — (middle) normalizes the dimensions of the layers and member parts of the layout component, making the visual representation noticeably more symmetrical and less accurate.
- **Fully Encrypted Layout Component** — (right-most) appears as a black box outline.



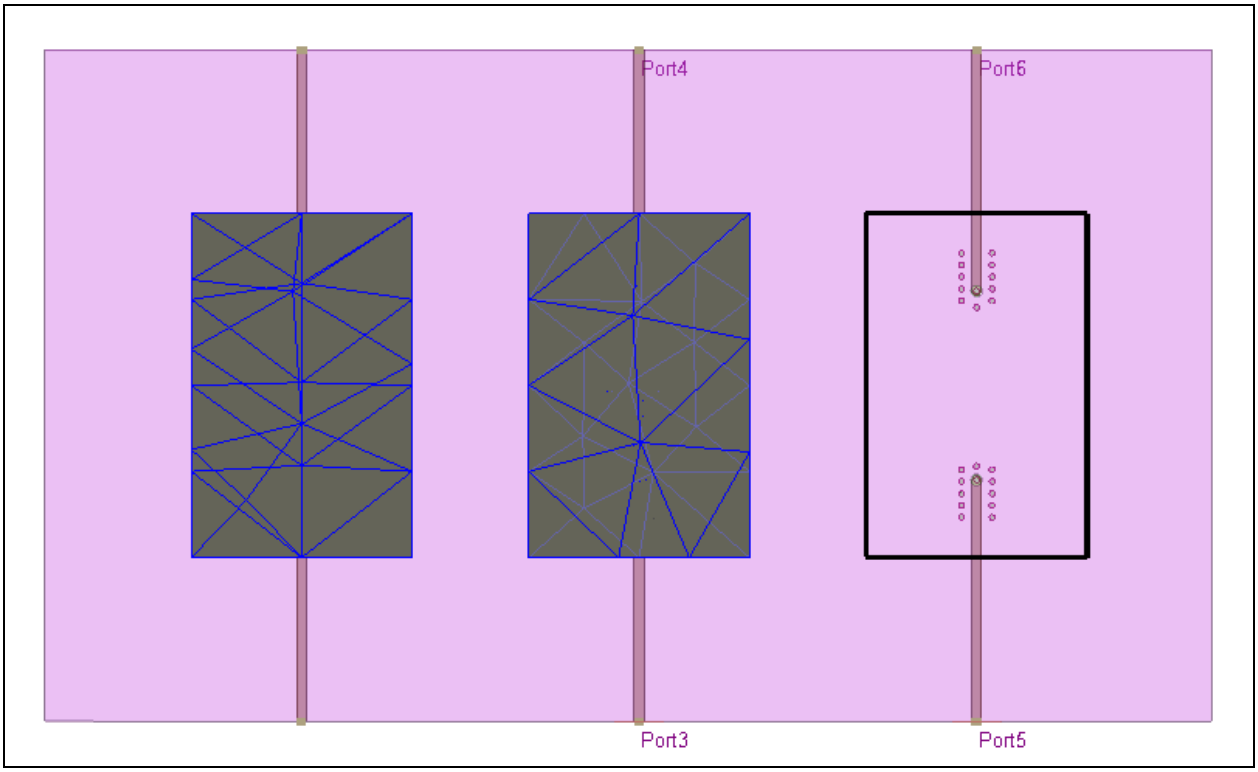
Layout Editor - Mesh View

Layout Component plotted meshes will differ greatly depending on their level of encryption. Referring to the following example:

- **Unencrypted Layout Component** — (left-most) represents the design accurately with a mesh as dense and complicated as the layout component requires depending on its materials, position, and dimensions
- **Encrypted Technology Layout Component** — (middle) normalizes the dimensions of the layout component and its member parts. Subsequently, the mesh is noticeably more symmetrical and less accurate
- **Fully Encrypted Layout Component** — (right-most) can not be assigned a mesh and will remain a black outline, or stencil. The only guidance the visual representation of a fully encrypted mesh gives users the accurate location of its ports.

Note: Layout Component members with different levels of encryption cannot be plotted together. Attempting to do so will result in an error message.

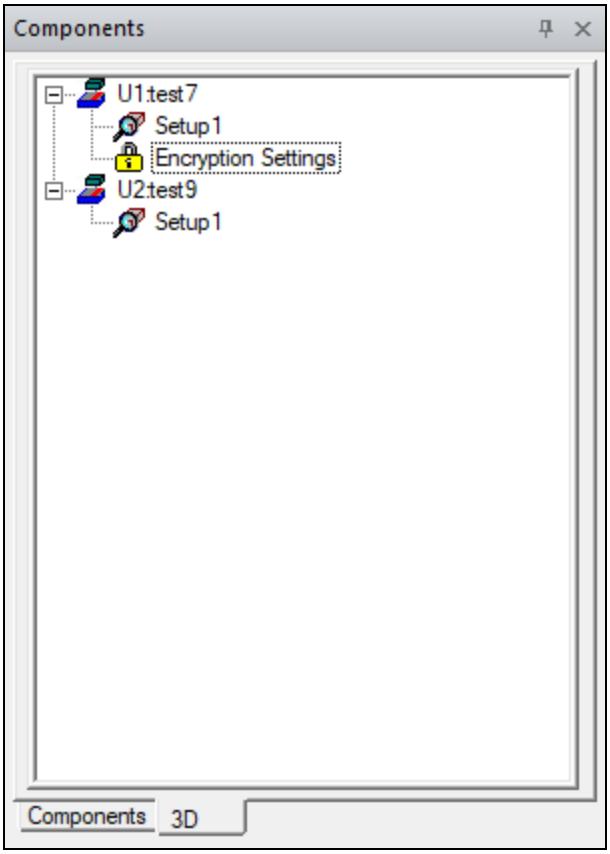
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Components Window

Refer to the **Components** window. A layout component that is ***not*** encrypted will list **Setup** mesh settings in an expandable dropdown menu. An encrypted layout component will have mesh settings and an **Encryption Settings** yellow lock icon.

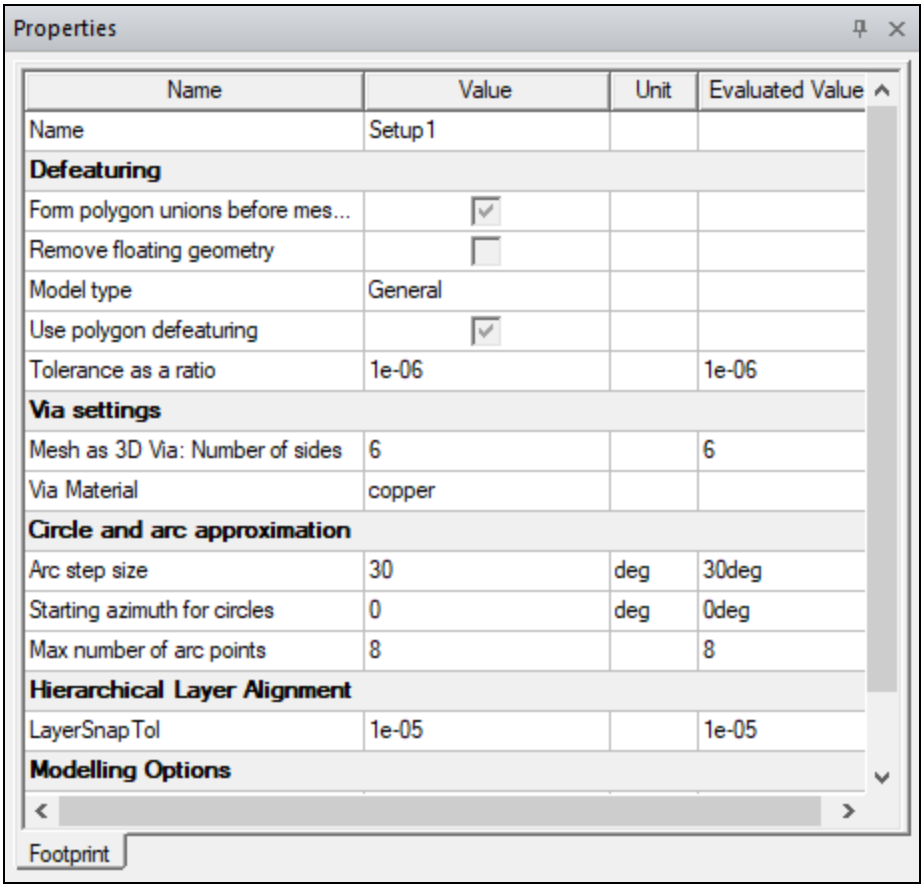
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



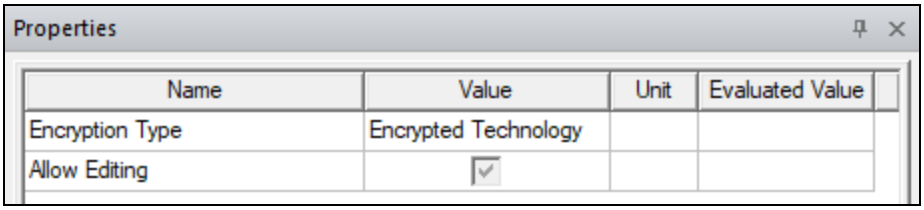
Properties Window

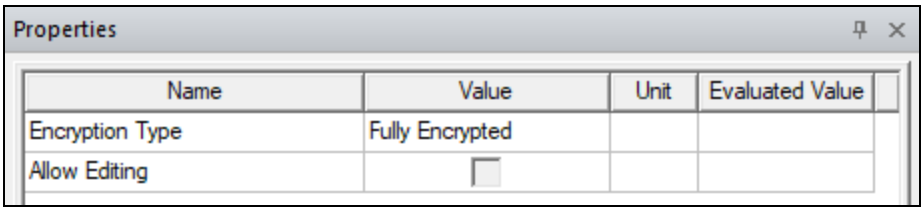
Select a **Setup** to view the layout component's accompanying mesh settings in the **Properties** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select the yellow lock icon to view the layout component's encryption settings in the **Properties** window.

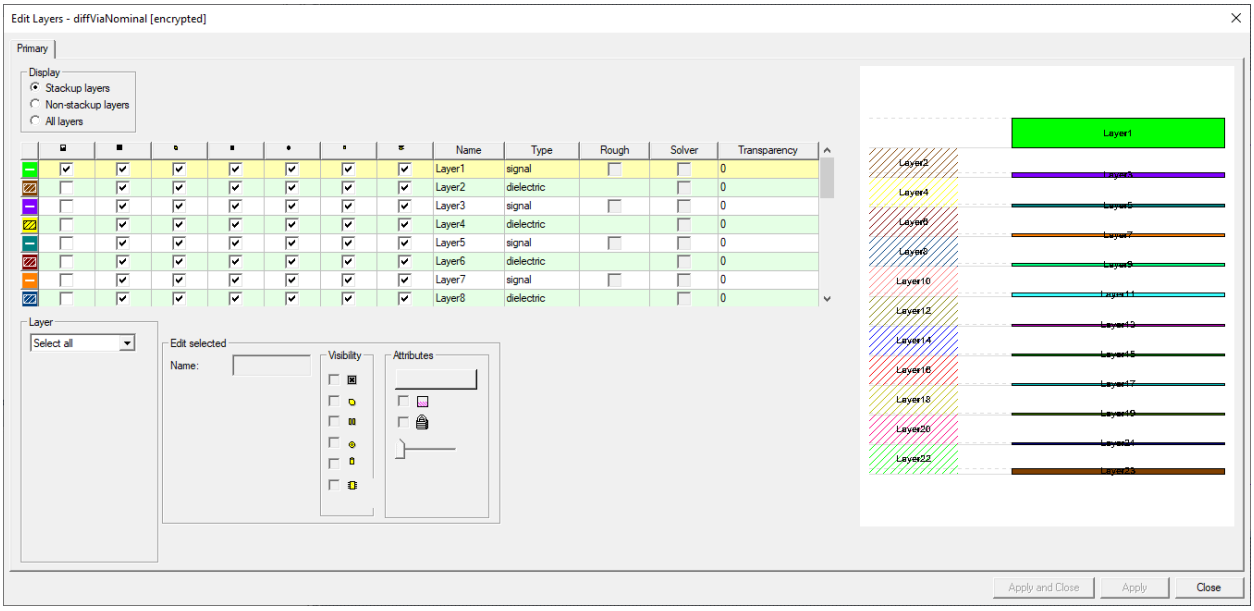




Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

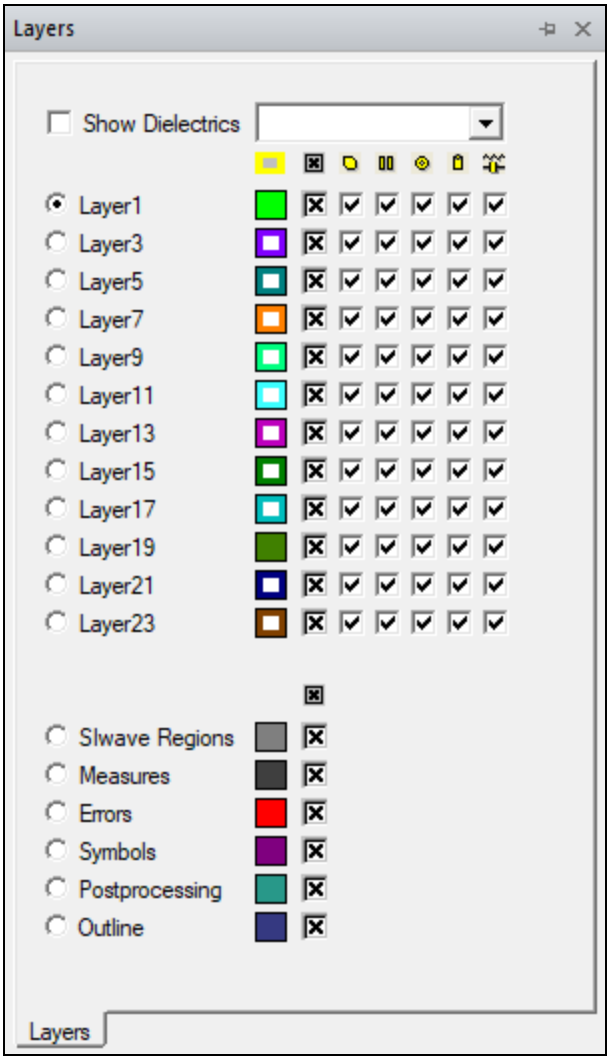
Edit Layers Window and Layers Pane

Navigate to the **Edit Layers** window. The layer **Names** have been obfuscated, and the physical parameters of the layers (e.g., material, dimensions, et cetera) have either been hidden or normalized.



The names of the layers have also been obfuscated on the **Layers** pane.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



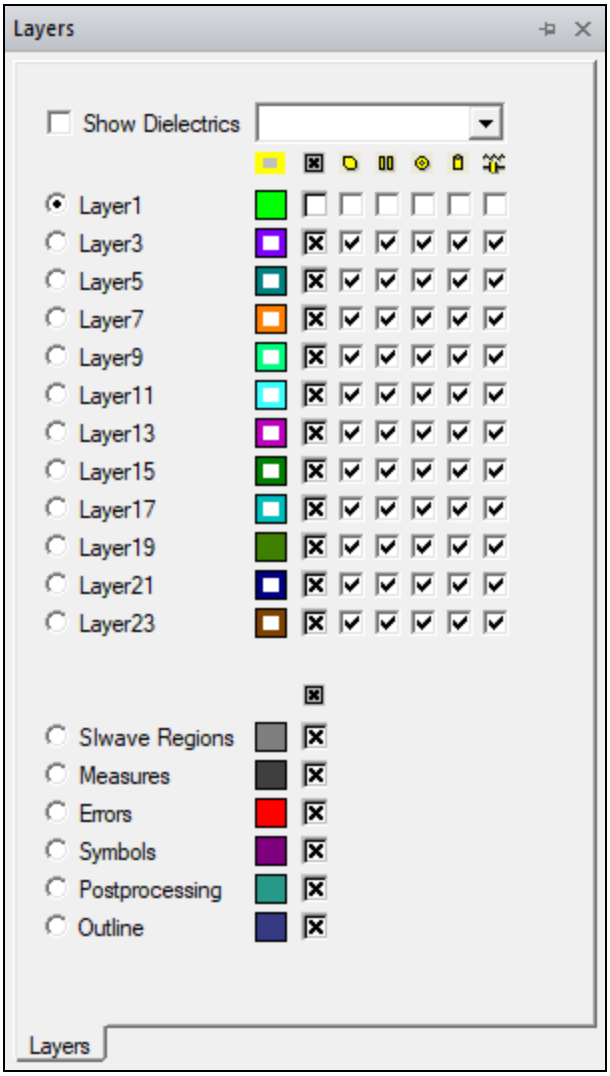
Hiding Layers and Nets

Encryption may make it difficult to view complex layout components in the **Layout Editor**, but nets and layers can be temporarily hidden to make it easier for users to view layout component members.

To Hide a Layer

From the **Layers** pane, remove the check on the box in the selected layer.

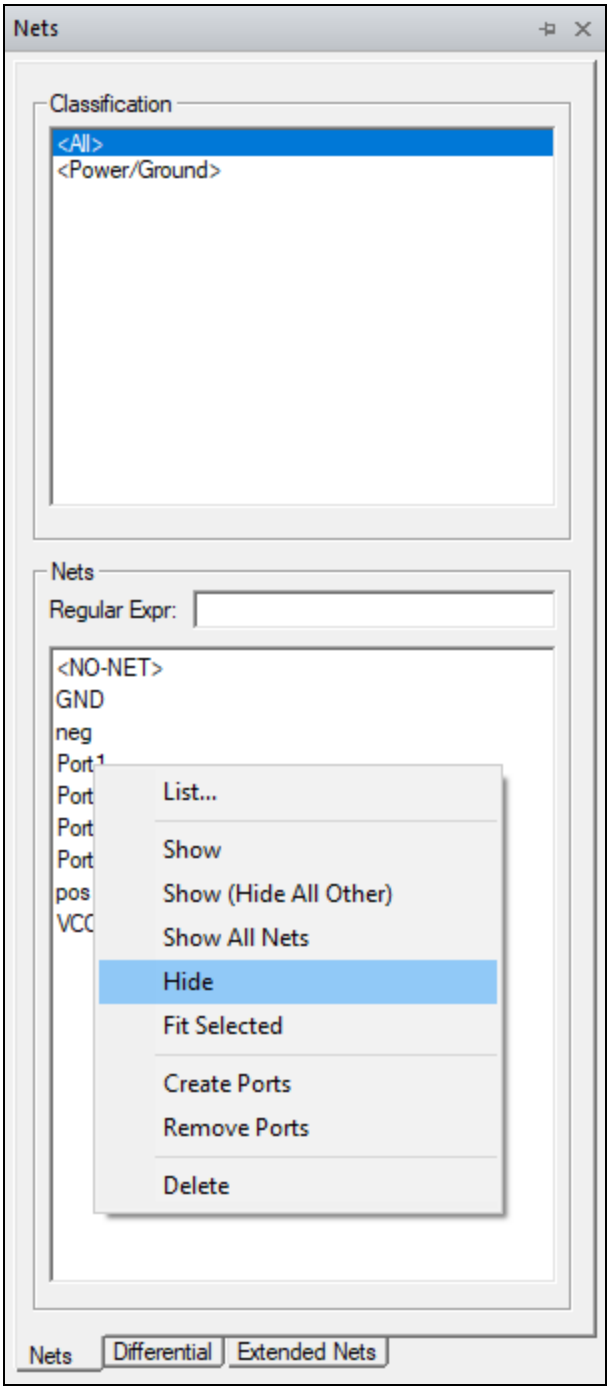
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To Hide a Net

From the **Nets** pane, right-click a net and select **Hide**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

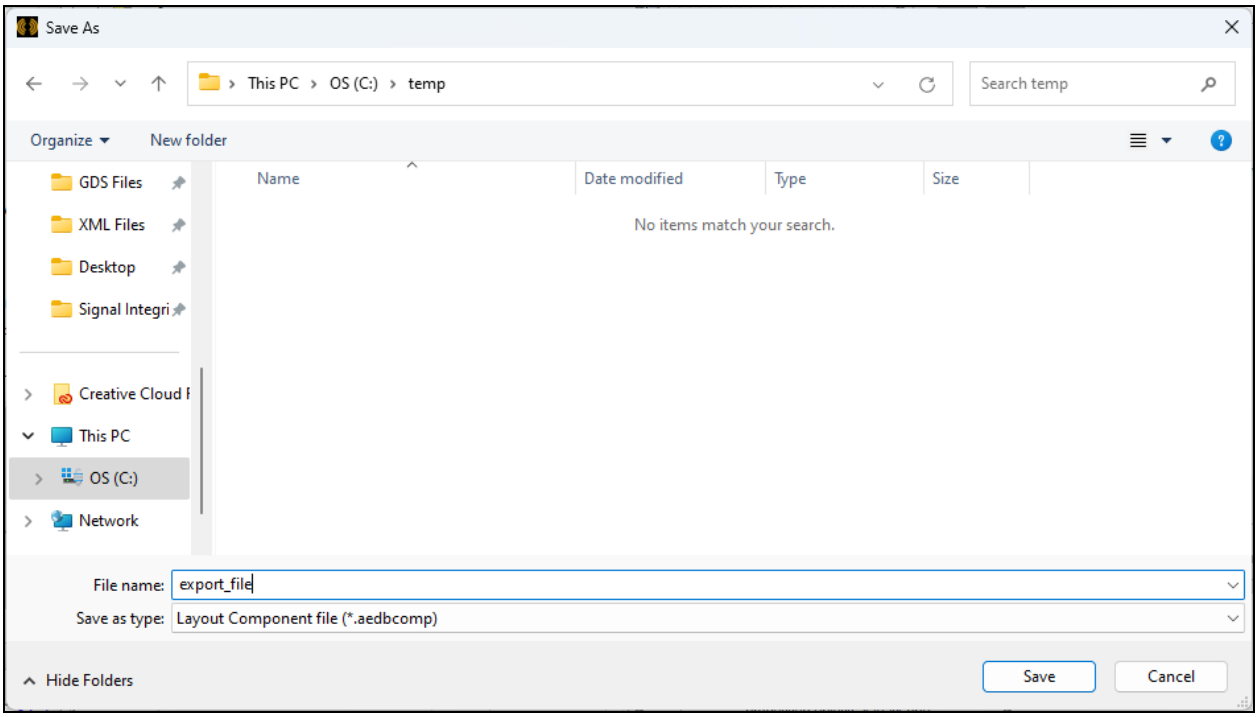


Exporting a Layout Component

Follow these steps to export a layout component from the Layout Editor.

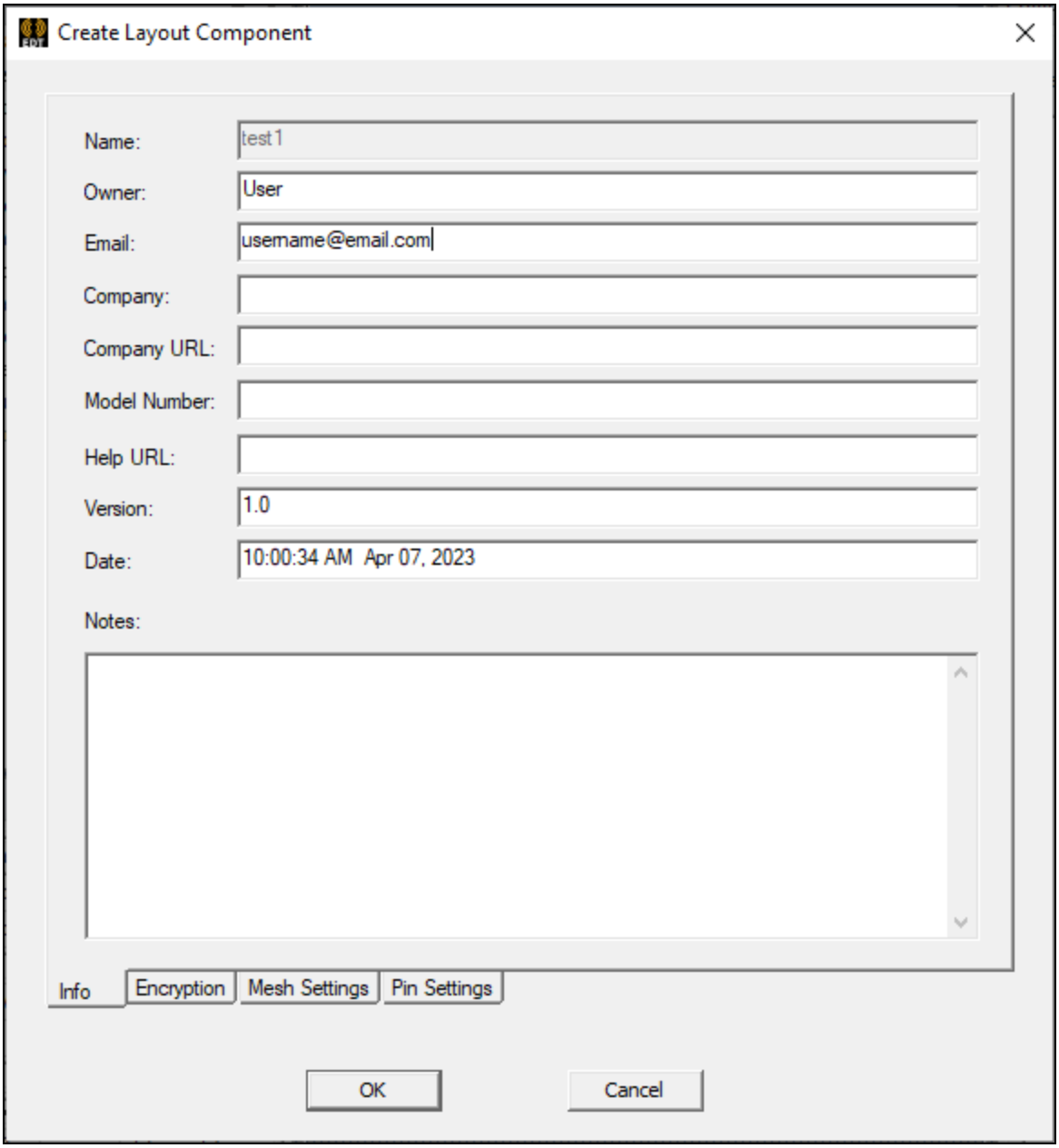
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Layout Editor**, select a layout component.
2. Depending on whether the Ansys Electronics Desktop (AEDT)™ electronics systems design platform is running in General mode or IC mode, navigate to one of the following selections to open an explorer window:
   - In General mode, navigate to **Layout** > **Export** > **Layout Component**.
   - In IC mode, navigate to **Layout** > **Export Layout Component**.
3. Navigate to an appropriate directory to save the exported layout component. Then enter a **File name** for the layout component and, if necessary, choose **Layout Component file (*.aedbcomp)** on the **Save as type** drop-down menu (selected by default).



4. Click **Save** to close the explorer window and open the **Create Layout Component** window. The **Create Layout Component** window allows users to add meta data, encryption, and mesh settings to an HFSS 3D Layout design layout component before exporting it.

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 **Info** tab, enter metadata into the appropriate fields. The layout component's file name is shown in the **Name** field.
6. Click the **Encryption** tab. Use the **Encryption** tab to protect the layout component with two levels of security. Encryption is a common technique for encoding information in a manner that is very difficult for unauthorized users to decode. When these options are enabled for layout components, the contents of the layout component file are encrypted to

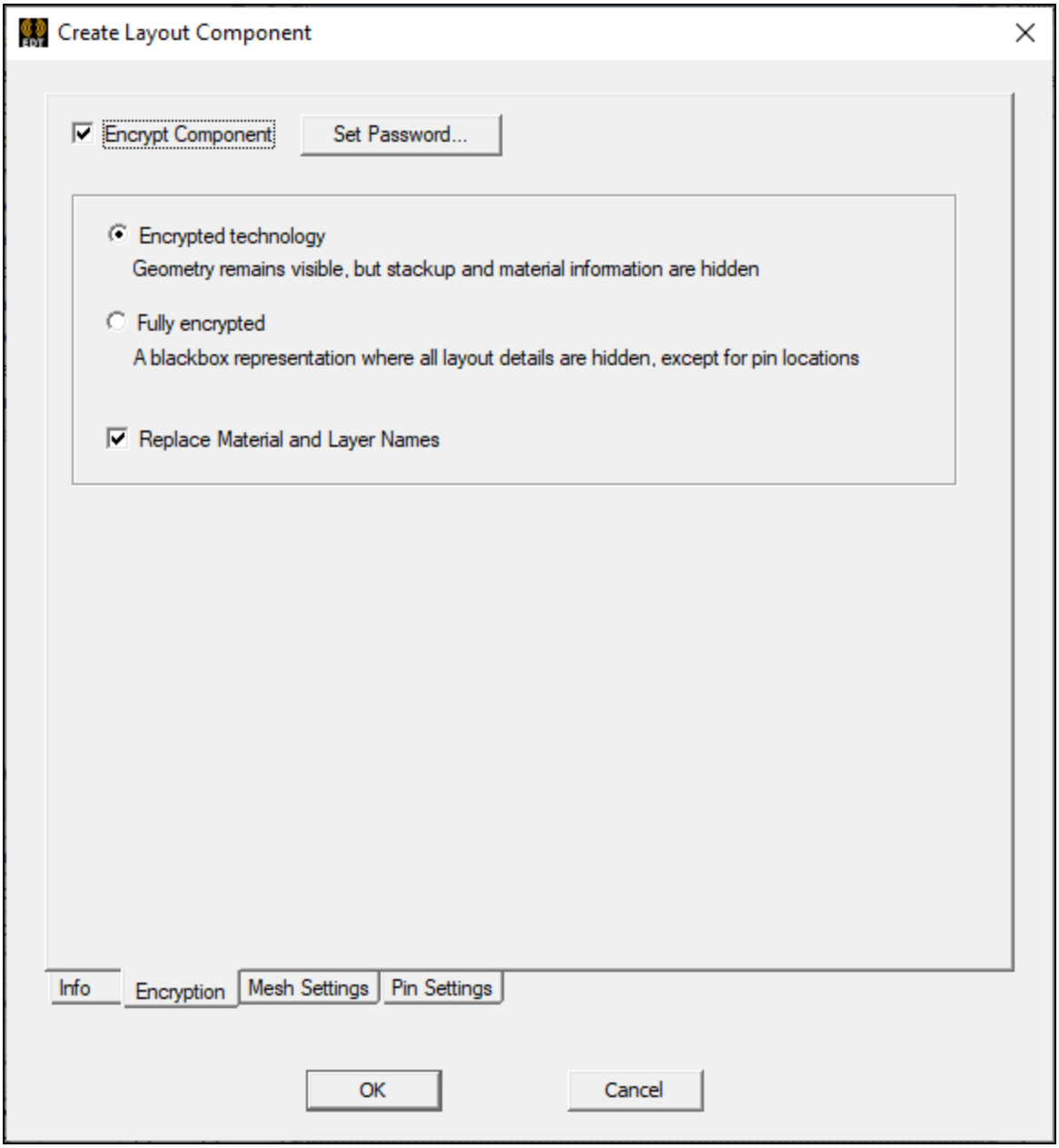
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

protect them from external viewing or editing. Layout Components use the Advanced Encryption Standard (AES) with a 256-bit key and password-based key derivation.

Note:

Once a layout component is encrypted and users click **OK**, encryption cannot be removed on the layout component. However, encryption strength can be increased (i.e., if the user chooses not to encrypt the layout component, the exported layout component can be re-imported and exported again as an **Encrypted technology** layout component. If the user exports the **Encrypted technology** layout component, the exported layout component can be re-imported and exported again as a **Fully encrypted** 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.

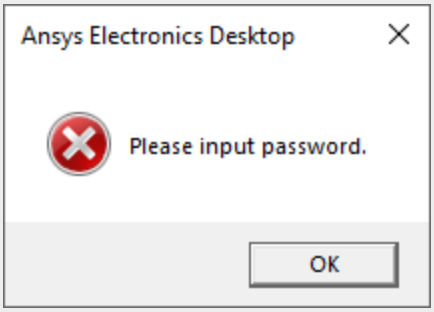


7. From the **Encryption** tab, choose from the following options:
   - No encryption – remove the check on the **Encrypt Component** box.
   - **Encrypted technology** – geometry will remain visible, but stackup and material information is encrypted. The layout component may be editable. Click **Set Password** to open the **Set Passwords** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

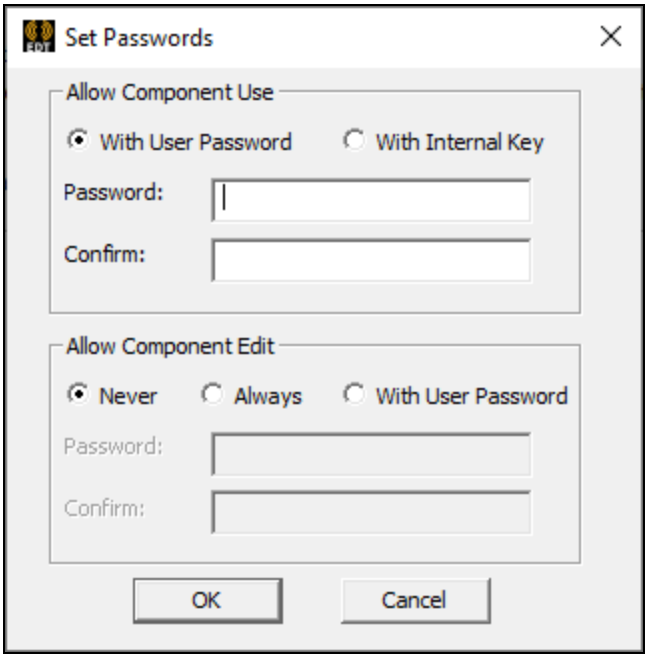
- **Fully encrypted** – all data, except port locations, is hidden under a blackbox stencil. The layout component will remain uneditable. Click **Set Password** to open the **Set Passwords** window.
- **Replace Material and Layer Names** – check the box to obfuscate all material and unique layer names (e.g., **Dialetric_1** will be renamed **Layer1** and the material column will be hidden from view).

  It is not possible to edit a **Fully encrypted** layout component unless you choose **Always** or **With User Password** in the **Allow Component Edit** area, when you define the layout component.

> **Note:**
>
> If there is a check in the **Encrypt Component** box, the **Set Passwords** window will open if users attempt to leave the **Encryption** tab. Attempting to bypass the window will produce a **Warning: Please input password.** dialog box. This is true even if no encryption is selected. To leave the **Encryption** tab, uncheck the **Encrypt Component** box.
>
> 
>

8. Click **OK** to open the **Set Password** window, enter and re-enter a password in the selected fields in the **Allow Component Use** area and decide whether future layout component editing 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.

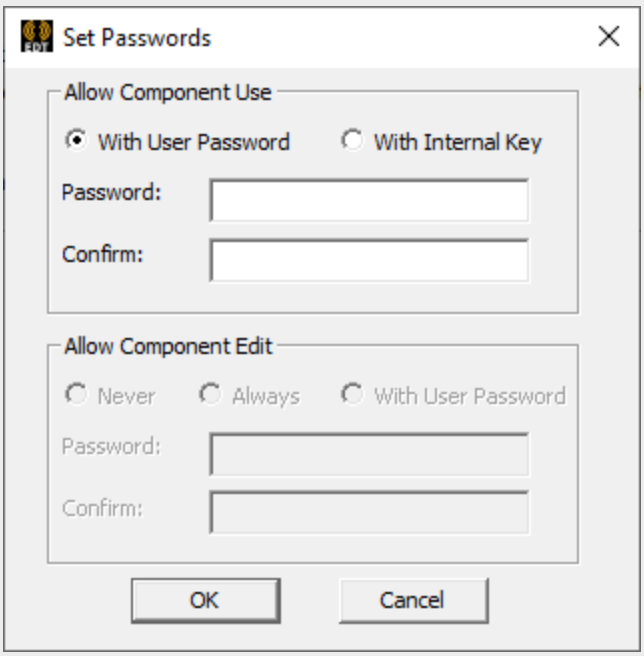


9. From the **Allow Component Use** area, choose from the following:
    - **With User Password** – the layout component can be exported as an *.aedbcomp* file and reopened in AEDT or other compatible software with the correct password.
    - **With Internal Key** – the layout component can be exported as an *.aedbcomp* file and reopened in (and only in) AEDT without a password.
10. If future layout component editing is required, from the **Allow Component Edit** area, choose from the following:
    - **Never** – once the **Set Passwords** window is closed, the layout component will no longer be editable.
    - **Always** – if future layout component editing is selected, and no further security is appropriate.
    - **With User Password** – if future layout component editing is selected, but some further security is appropriate. Enter and re-enter a password in the selected fields.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

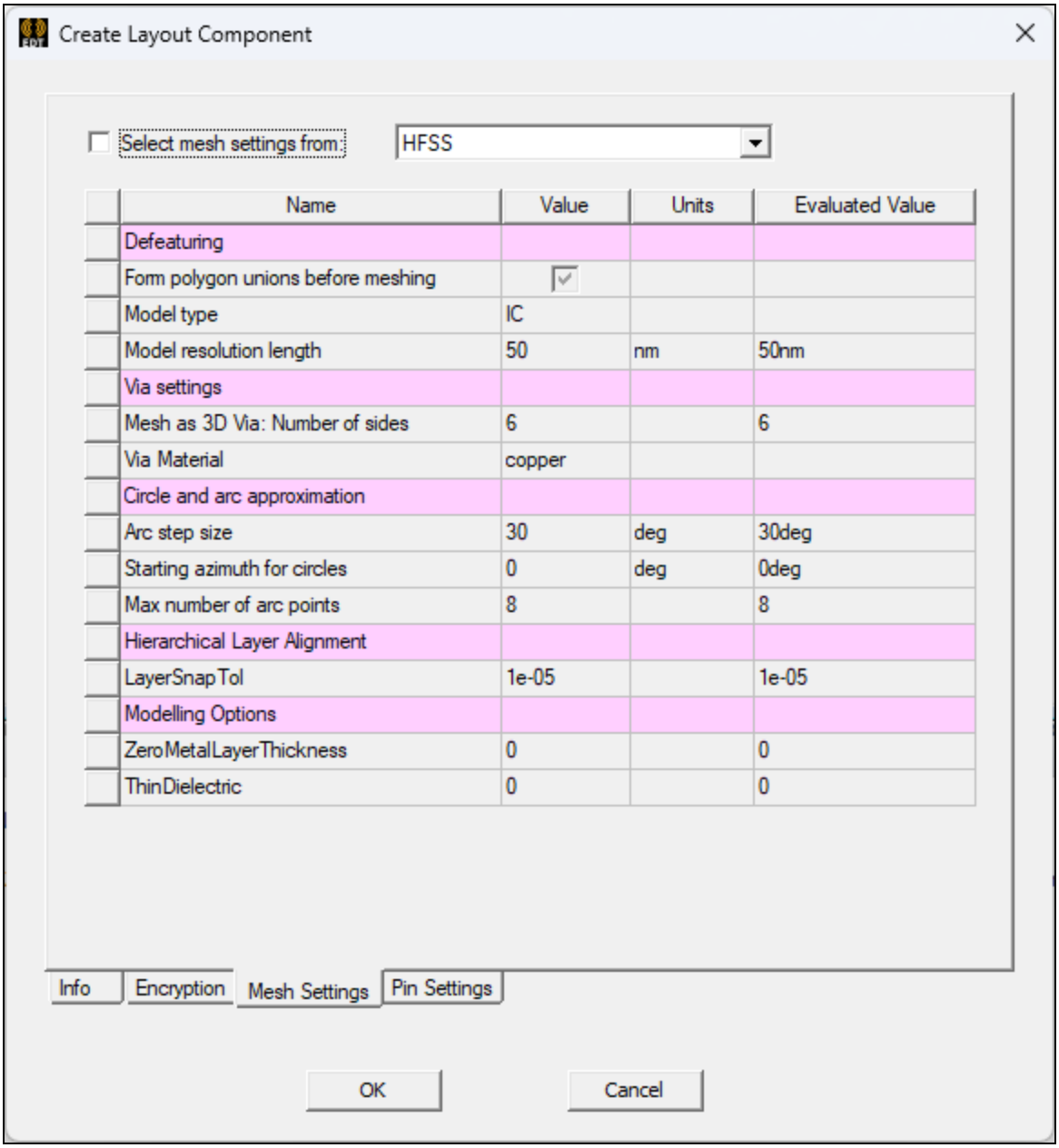
Note:

Choosing **Fully encrypted**deactivates the **Allow Component Edit** area. This level of encryption is a permanent state and the layout component will not be editable after the **Create Layout Component** window is finally closed.



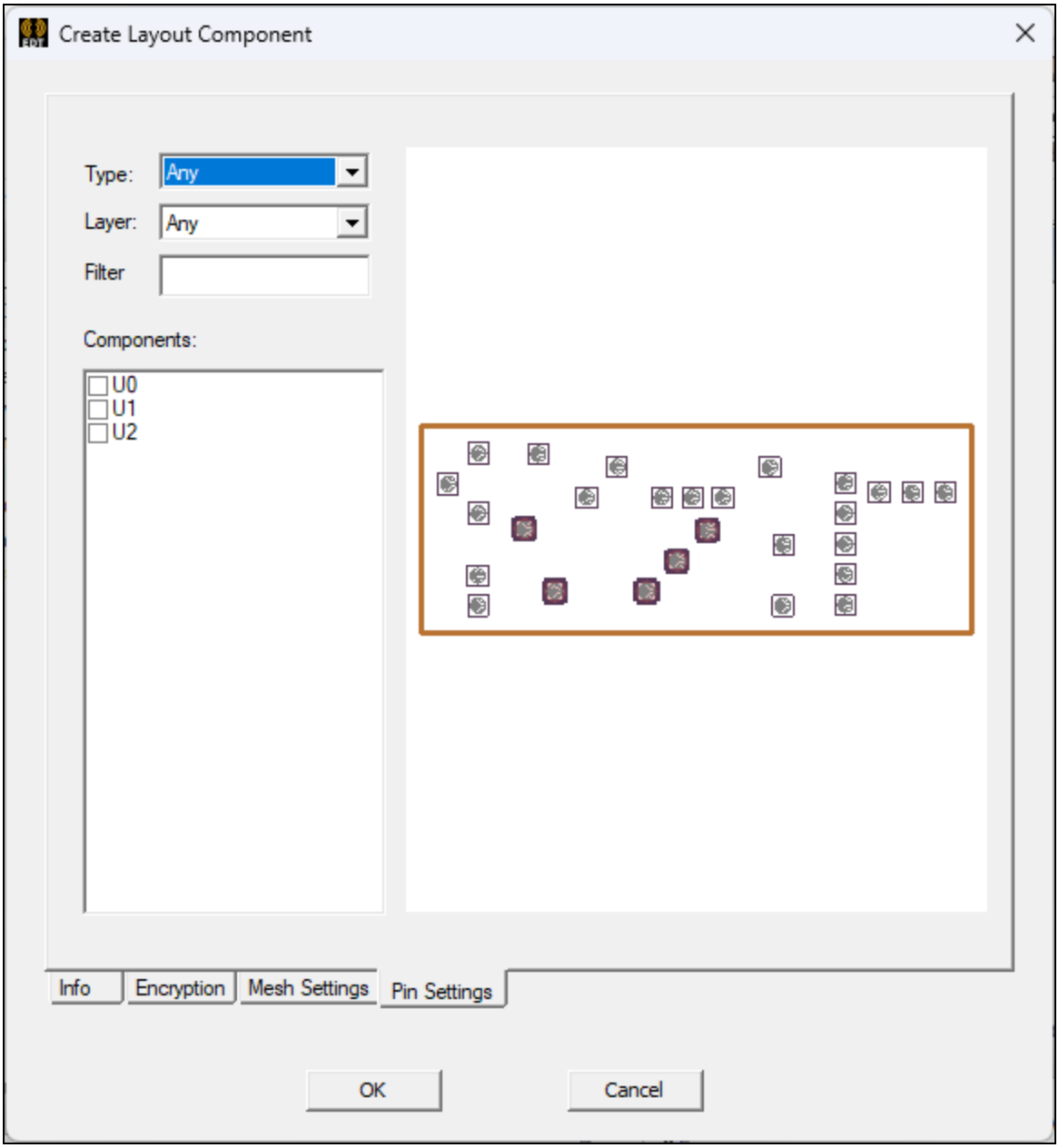
11. Click **OK** to save, close the **Set Passwords** window, and return to the **Create Component** window.
12. Click the **Mesh Settings** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



13. To save existing mesh settings with the exported layout component, check the **Select mesh settings from** box and make a selection from the drop-down menu.
14. If **Fully encrypted** is selected on the **Encryption** tab, click the **Pin Settings** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



15. From the **Pin Settings** tab, choose from the following:

    - Use the **Type** and **Layer** drop-down menus, as well as manually entering **Filter** variables to filter the Layout Component list of available parts selected on the **Layout Editor**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Components** — check boxes adjacent to each layout component part to leave the layout component(s) exposed, despite full encryption. The exposed layout components are shown in preview window.

16. Click **OK** to export the layout component.
17. After the layout component is configured, click **OK** to add it to the active design.

Editing a Layout Component Definition

The **Edit Definition** feature allows users to create a new layout component definition using an existing layout component file. This speeds up the task of creating new versions of layout components without having to start on the beginning.

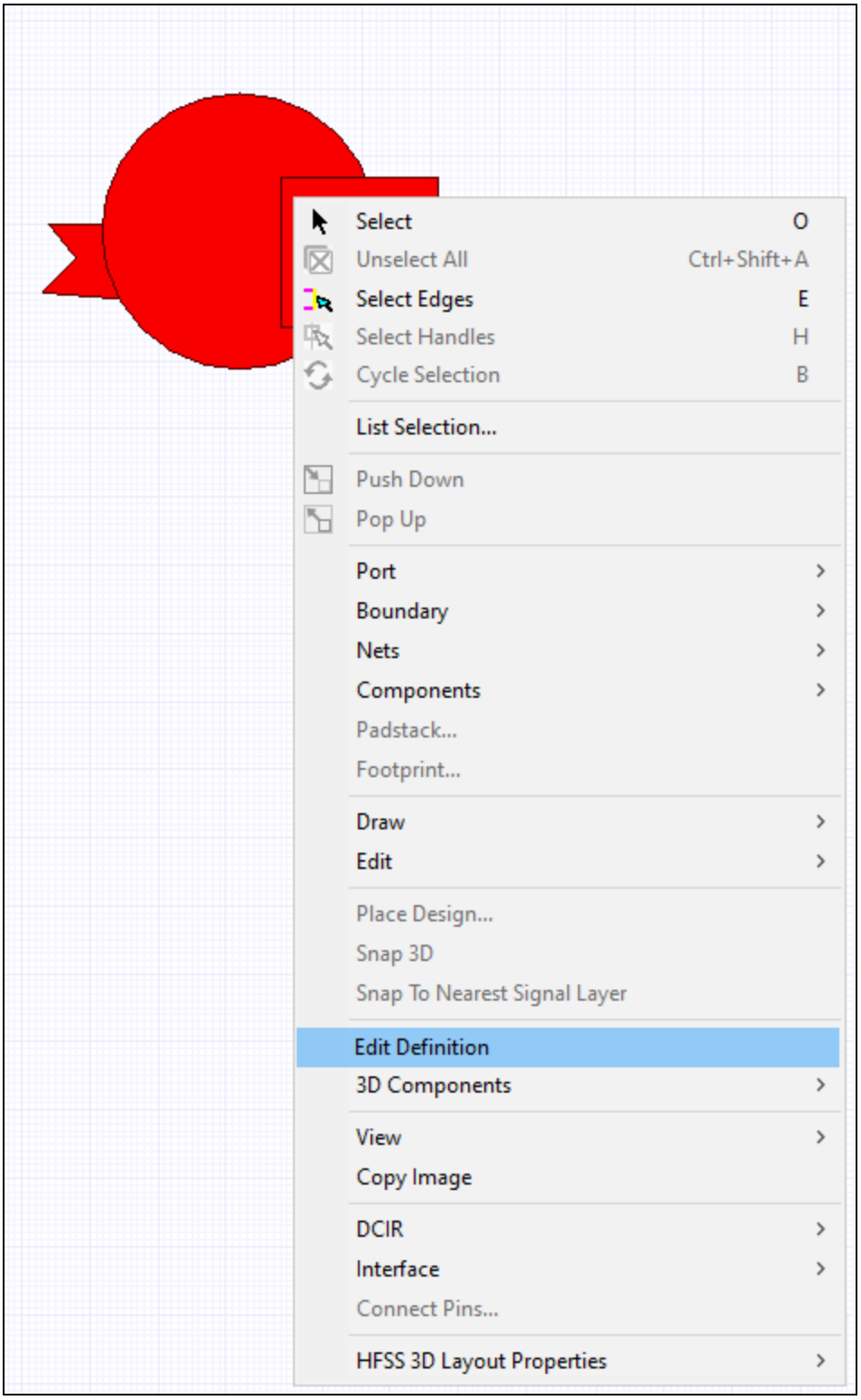
> **Important:**
>
> Fully encrypted layout components can not be edited as described in this section.

Follow these steps to edit a layout component definition.

1. Navigate to the **Edit Definition** window via one of the following paths:

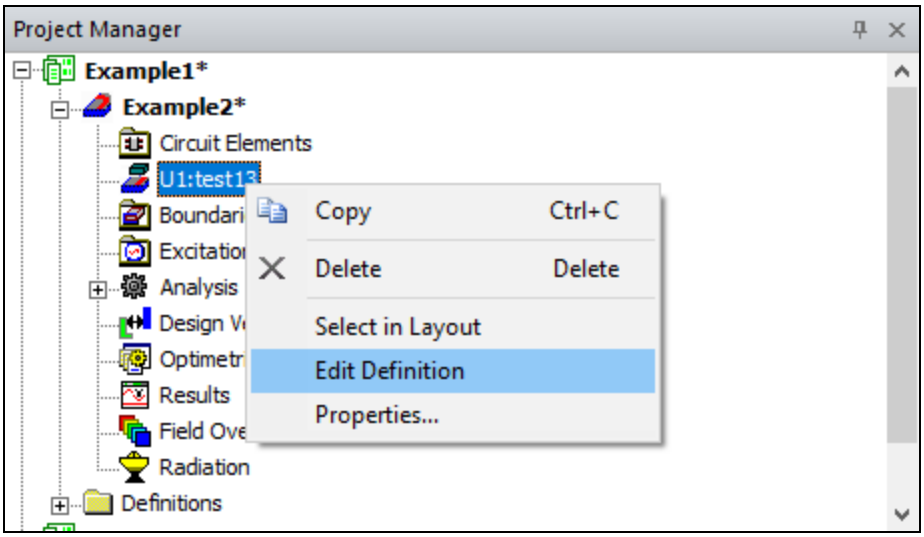
- From the **Layout Editor**, right-click the applicable, layout component and select **Edit Definition**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

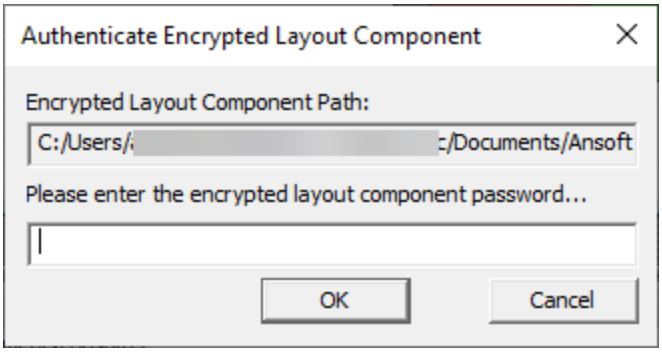


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click the applicable, layout component and select **Edit Definition**.



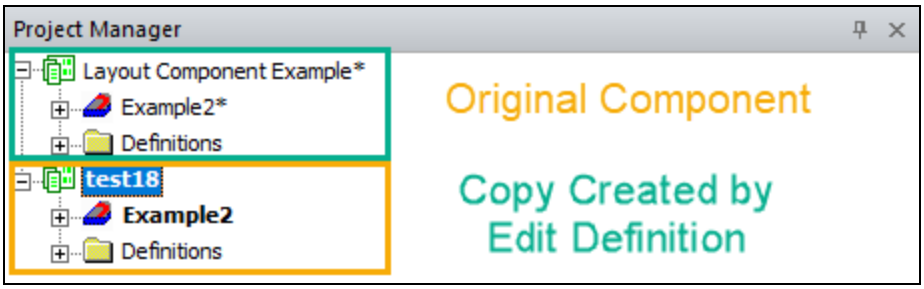
2. If the layout component is encrypted, an **Authenticate Encrypted Layout Component** window appears. Enter the layout component's encryption password, then click **OK**.



3. A duplicate project is created. Copies of the original layout component's 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 layout component; design data is added to the applicable, folders in the **Project Manager**. A copy of the layout component opens in the **Layout Editor**, and a new project folder 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.

containing a copy of the original design.

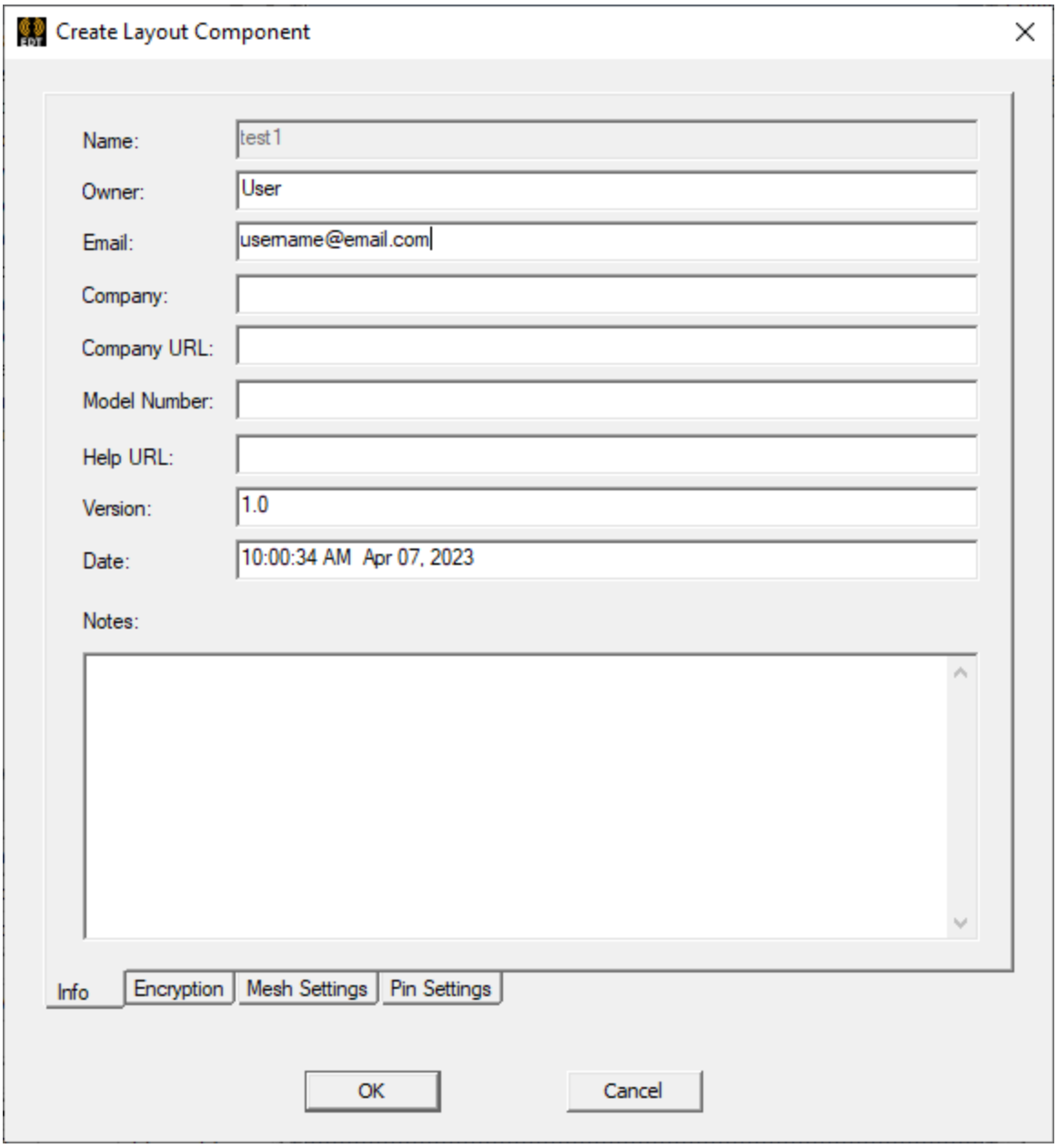


4. Edit the layout component, as preferred (refer to Components).
5. To save the edited layout component, do one of the following:

- To save the edited layout component as its own layout component file (*.*aedbcomp*), click **File** > **Save As** to open an explorer window. Navigate to the applicable, directory, enter a **File name:** for the layout component, and click **Save** to close the explorer window. The procedure is complete. The layout component will be saved with the original layout component's settings and encryption.
- To save the edited layout component in the current project, click **Save** to open the **Create**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layout Component window. Then proceed to **step 6**.



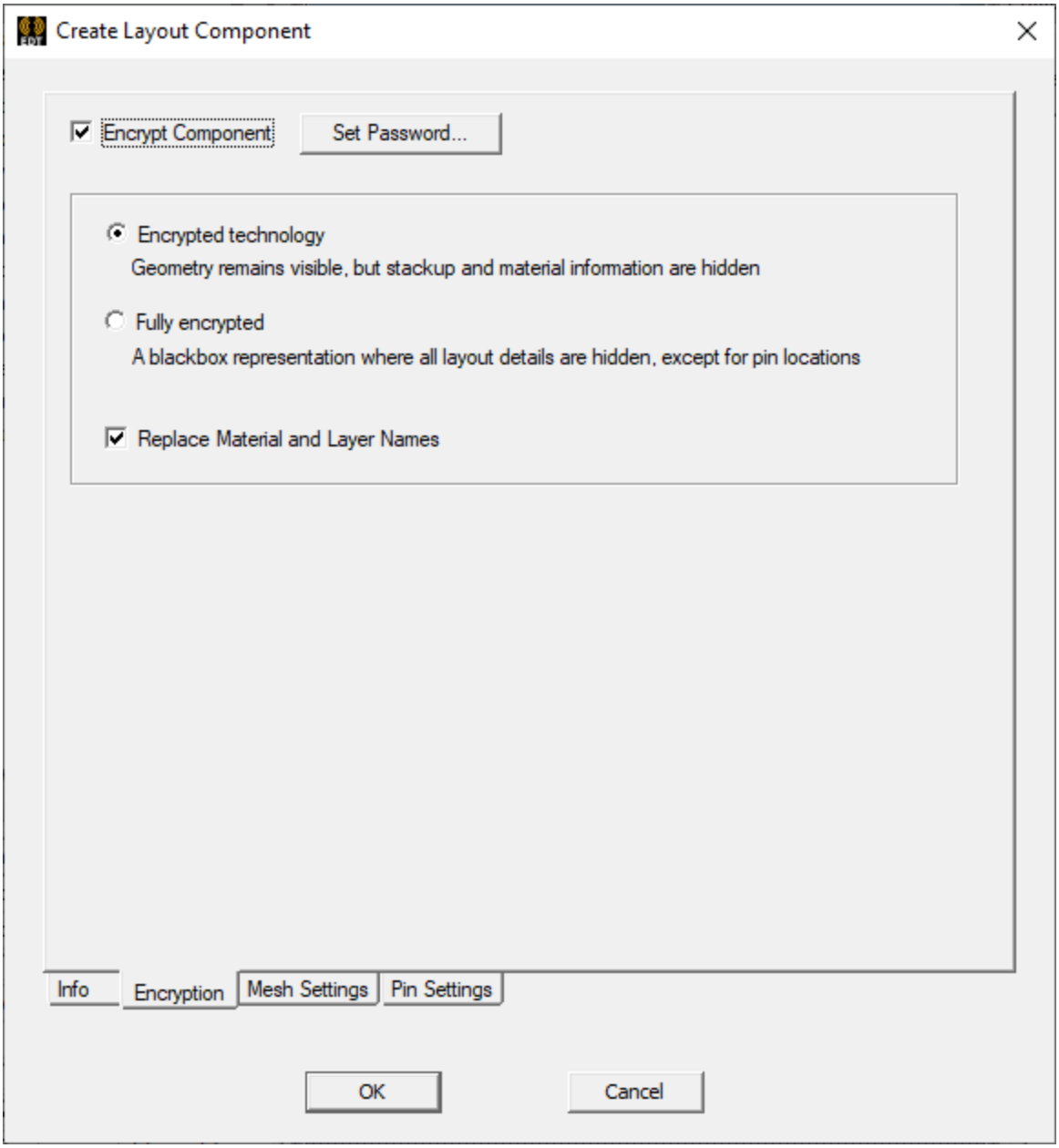
6. From the **Info** tab, enter metadata into the appropriate fields. The layout component's file name is shown in the **Name** field.
7. Click the **Encryption** tab. Use the **Encryption** tab to protect the layout component with two levels of security. Encryption is a common technique for encoding information in a manner that is very difficult for unauthorized users to decode. When these options are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enabled for layout components, the contents of the layout component file are encrypted to protect them from external viewing or editing. Layout Components use the Advanced Encryption Standard (AES) with a 256-bit key and password-based key derivation.

> **Note:**
>
> Once a layout component is encrypted and users click **OK**, encryption cannot be removed on the layout component. However, encryption strength can be increased (i.e., if the user chooses not to encrypt the layout component, the exported layout component can be re-imported and exported again as an **Encrypted technology** layout component. If the user exports the **Encrypted technology** layout component, the exported layout component can be re-imported and exported again as a **Fully encrypted** 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.

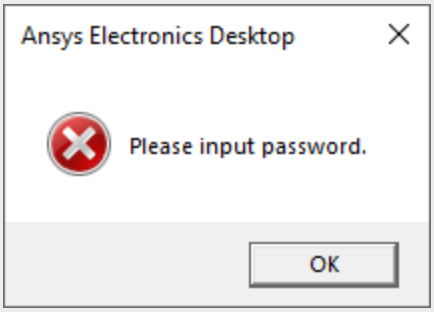


8. From the **Encryption** tab, choose from the following options:
    - No encryption – remove the check on the **Encrypt Component** box.
    - **Encrypted technology** – geometry will remain visible, but stackup and material information is encrypted. The layout component may be editable. Click **Set Password** to open the **Set Passwords** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

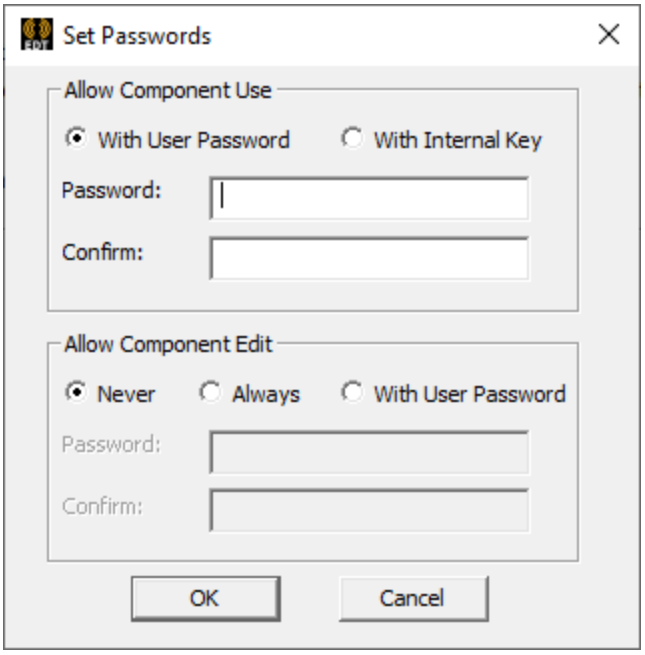
- **Fully encrypted** – all data, except port locations, is hidden under a blackbox stencil. The layout component will remain uneditable. Click **Set Password** to open the **Set Passwords** window.
- **Replace Material and Layer Names** – check the box to obfuscate all material and unique layer names (e.g., **Dialetric_1** will be renamed **Layer1** and the material column will be hidden from view).

  It is not possible to edit a **Fully encrypted** layout component unless you choose **Always** or **With User Password** in the **Allow Component Edit** area, when you define the layout component.

> **Note:**
>
> If there is a check in the **Encrypt Component** box, the **Set Passwords** window will open if users attempt to leave the **Encryption** tab. Attempting to bypass the window will produce a **Warning: Please input password.** dialog box. This is true even if no encryption is applicable,. To leave the **Encryption** tab, uncheck the **Encrypt Component** box.
>
> 
>

9. Click **OK** to open the **Set Password** window, enter and re-enter a password in the applicable, fields in the **Allow Component Use** area and decide whether future layout component editing is applicable,.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

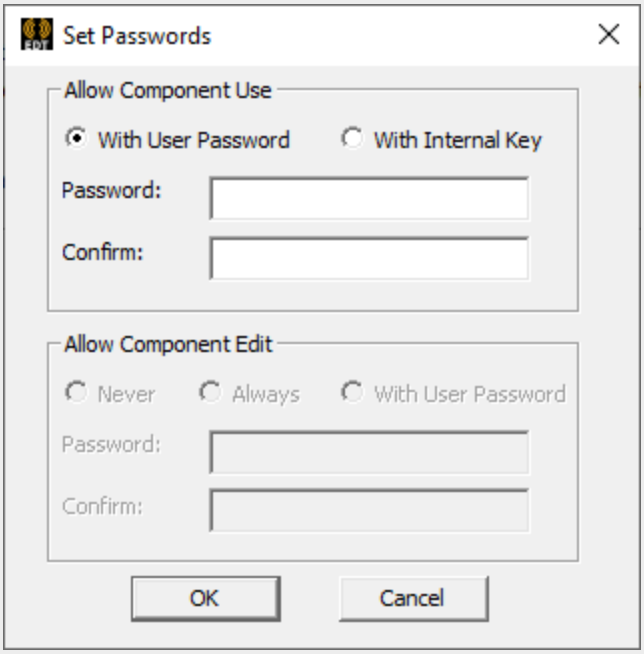


10. From the **Allow Component Use** area, choose from the following:
    - **With User Password** – the layout component can be exported as an *.aedbcomp* file and reopened in AEDT or other compatible software with the correct password.
    - **With Internal Key** – the layout component can be exported as an *.aedbcomp* file and reopened in (and only in) AEDT without a password.
11. If future layout component editing is required, from the **Allow Component Edit** area, choose from the following:
    - **Never** – once the **Set Passwords** window is closed, the layout component will no longer be editable.
    - **Always** – if future layout component editing is applicable,, and no further security is appropriate.
    - **With User Password** – if future layout component editing is applicable,, but some further security is appropriate. Enter and re-enter a password in the applicable, fields.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Choosing **Fully encrypted**deactivates the **Allow Component Edit** area. This level of encryption is a permanent state and the layout component will not be editable after the **Create Component** window is finally closed.



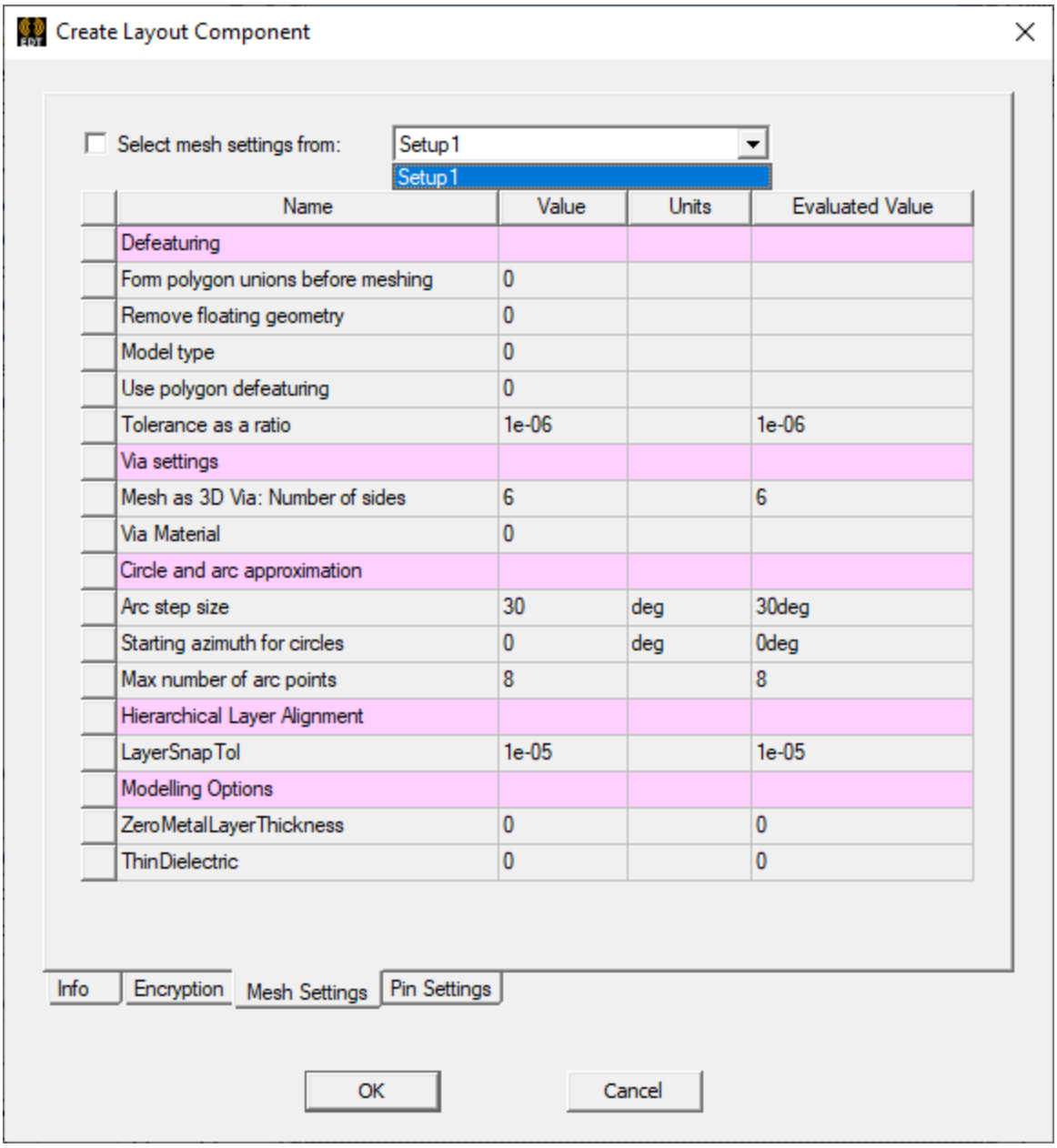
12. Click **OK** to save, close the **Set Passwords** window, and return to the **Create Component** window.

Note:

Once a layout component is encrypted and users click **OK**, encryption cannot be removed on the layout component. However, encryption strength can be increased (i.e., if the user chooses not to encrypt the layout component, the exported layout component can be re-imported and exported again as an **Encrypted technology** layout component. If the user exports the **Encrypted technology** layout component, the exported layout component can be re-imported and exported again as a **Fully encrypted** 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.

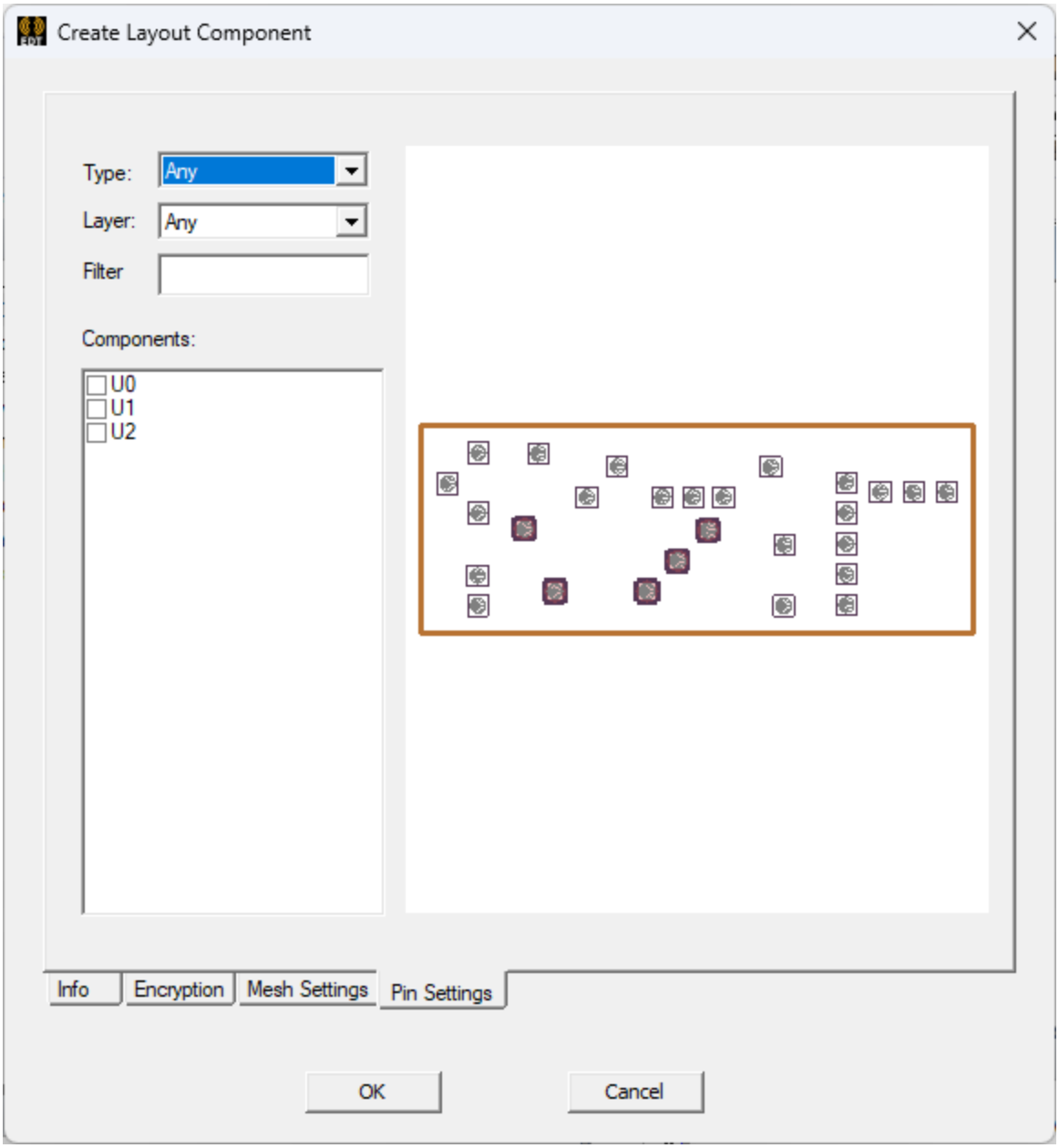
13. Click the **Mesh Settings** tab.



14. To save existing mesh settings with the exported layout component, check the **Select mesh settings from** box and make a selection from the drop-down menu.

15. If **Fully encrypted** is applicable, on the **Encryption** tab, click the **Pin Settings** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



16. From the **Pin Settings** tab, choose from the following:

    - Use the **Type** and **Layer** drop-down menus, as well as manually entering **Filter** variables to filter the Layout Component list of available parts applicable, on the **Layout Editor**.
    - **Components** — check boxes adjacent to each layout component part to leave the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

layout component(s) exposed, despite full encryption. The exposed layout components are shown in preview window.

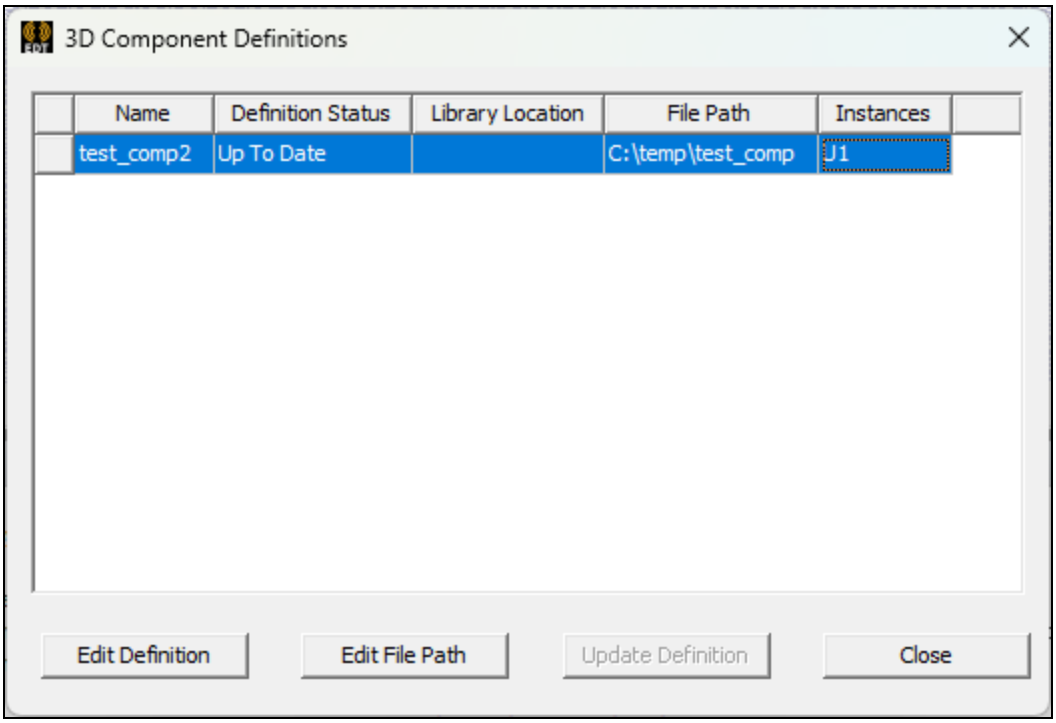
17. Click **OK** to export the layout component.
18. After the layout component is configured, click **OK** to add it to the active design.

The procedure is complete.

Replacing Layout Components

Layout components (and/or 3D components) can be replaced everywhere they have been placed in a design simultaneously from the **3D Component Definitions** window. Follow 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 applicable, 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All ports previously added to the instance(s) are also retained.
- Names are not retained.
- The replacement is irreversible.

Encrypted Designs (IC Mode)

Encryption obfuscates parameters, materials, and restricts access, however there are still many options available for users working with projects that include encrypted design, including:

- Adding layers to an encrypted design
- Trimming layers (i.e., deleting certain layers) from an encrypted design
- Enabling Solver Options, including surface roughness, in encrypted layers

For general guidance, refer to:

- Encrypted Layout Components
- Applying Encrypted Technology Files
- Encrypted Libraries

Adding Layers to an Encrypted Design

When a design is encrypted, users can add a signal or dielectric layer, and appropriate parameters (i.e., **Thickness** and **Material**) to the top or bottom of the encrypted stack without removing the encryption.

> **Note:**
>
> The stackup remains encrypted. After a layer is added, it will conform to the existing encryption level.

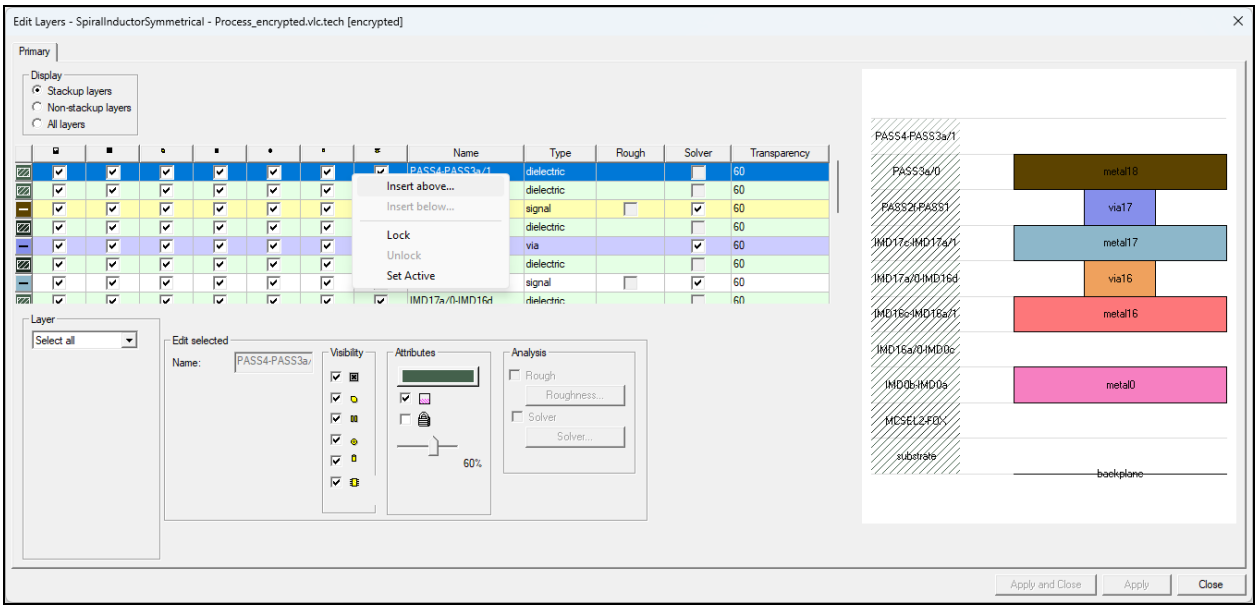
Follow these steps to add a layer to an encrypted stackup. Refer to the Edit Layers window.

1. To open the **Edit Layers** window, do one of the following from the **Layout** tab.

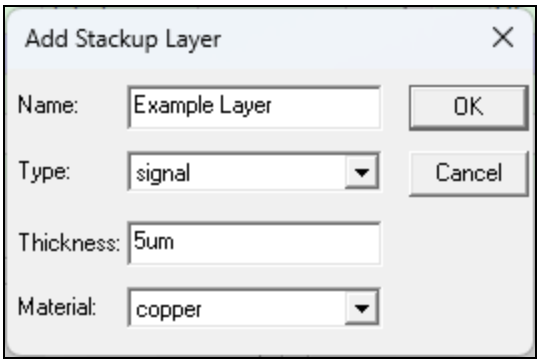
- From the **Layout** ribbon, select **Layers dialog**.
- From **Layout** or **Footprint**, select **Layers**.
- Right-click in the **Layout Editor** and select **Edit Layers**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the **Grid Control Table**, right-click in the top or bottom row and select **Insert above/Insert below** to open the **Add Stackup Layer** window.

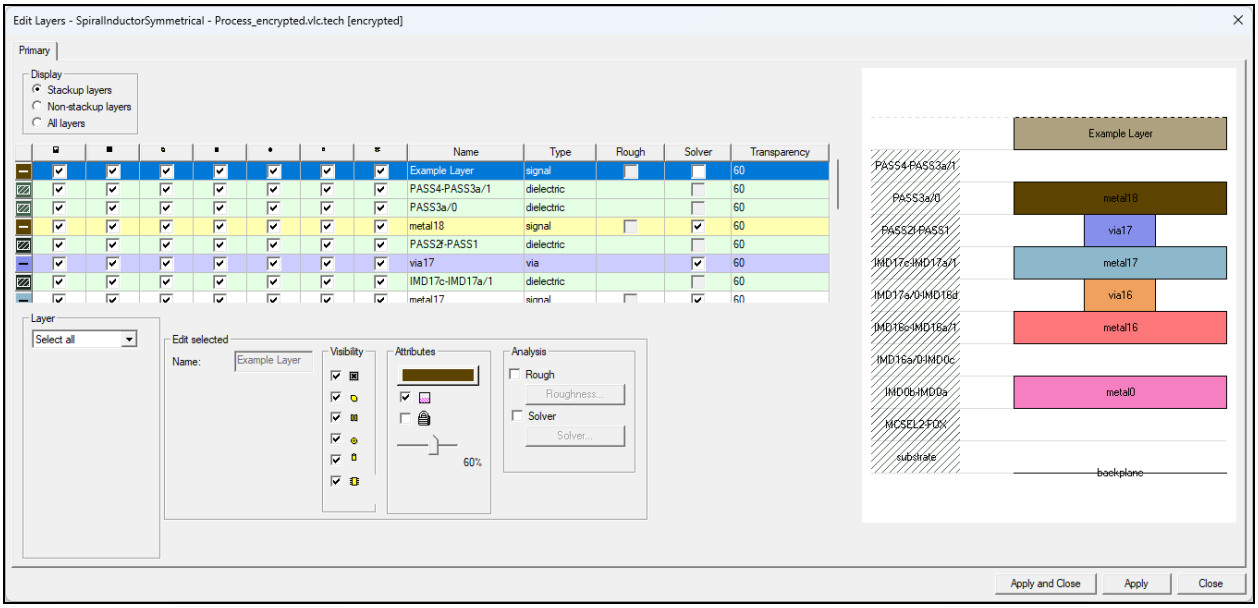


3. From the **Add Stackup Layer** window, enter an appropriate **Name**, **Type** (i.e., **signal** or **dielectric**), **Thickness** (e.g., **5um**), and **Material** from the drop-down menu (e.g., **copper**).



4. Click **OK** to close the **Add Stackup Layer** window and add the new layer to the **Grid Control Table**.

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 remove layers from an encrypted design, refer to Trimming Layers from an Encrypted Design.

Enabling Solver Options in Encrypted Layers

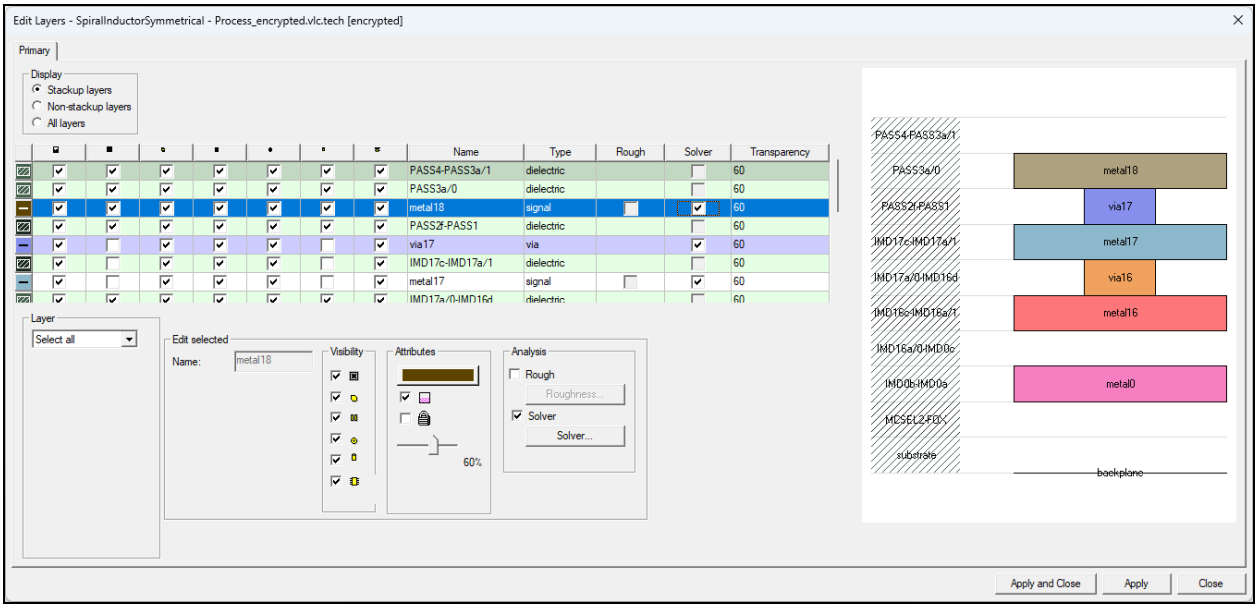
Follow these steps to enable solver options and surface roughness in an encrypted stackup.

1. To open the **Edit Layers** window, do one of the following from the **Layout** tab.

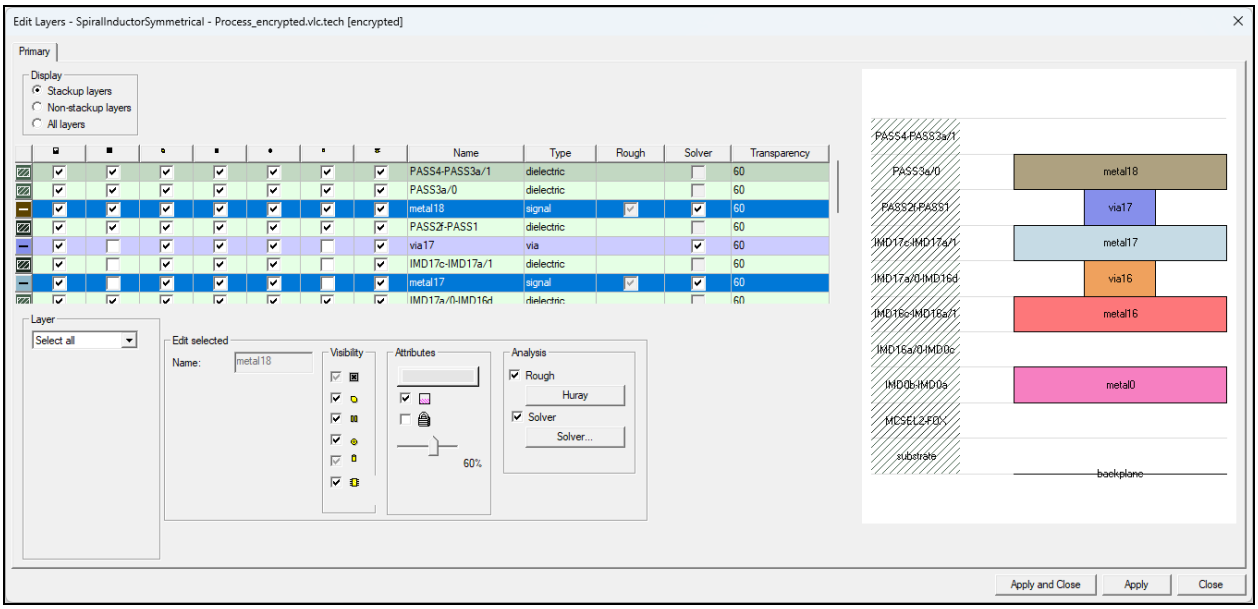
   - From the **Layout** ribbon, select **Layers dialog**.
   - From the **Layout** or **Footprint** menu, select **Layers** .
   - Right-click in the **Layout Editor** and select **Edit Layers**.

2. From the **Grid Control Table**, check any/all of the active boxes in the **Solver** column, or select one or more layers and check the **Solver** box in the **Analysis** area.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

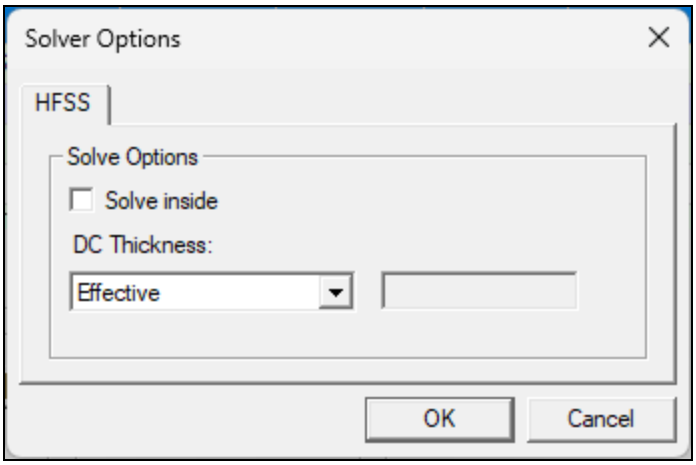


3. By default, layers do not have surface roughness. If applicable, add **Rough** status to one or more signal layers by selecting the layer(s) and checking the **Rough** box in the **Analysis** area. The roughness is added to the layer when its mesh is created. Refer to Setting the Layer Surface Roughness.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Checking either **Solver** boxes in the **Grid Control Table** or the **Solver** box in the **Analysis** area activates the **Solver** button. Click **Solver** to open the Solver Options window.



5. From the **Solver Options** window, do the following:

   - **Solve inside** check box — check to enable all objects in the layer to have the "solver inside" attribute . It is possible to perform solve inside on an object-by-object basis. Refer to HFSS 3D Layout Properties
   - **DC Thickness** drop-down menu — controls how DC thickness is calculated for the conductor lumps on the layer:
     - **Effective** — calculates thickness as twice the volume/surface area.
     - **Layer** — uses the layer's thickness.
     - **Manual** — enter a value in the field.

Creating Instance Arrays in IC Mode

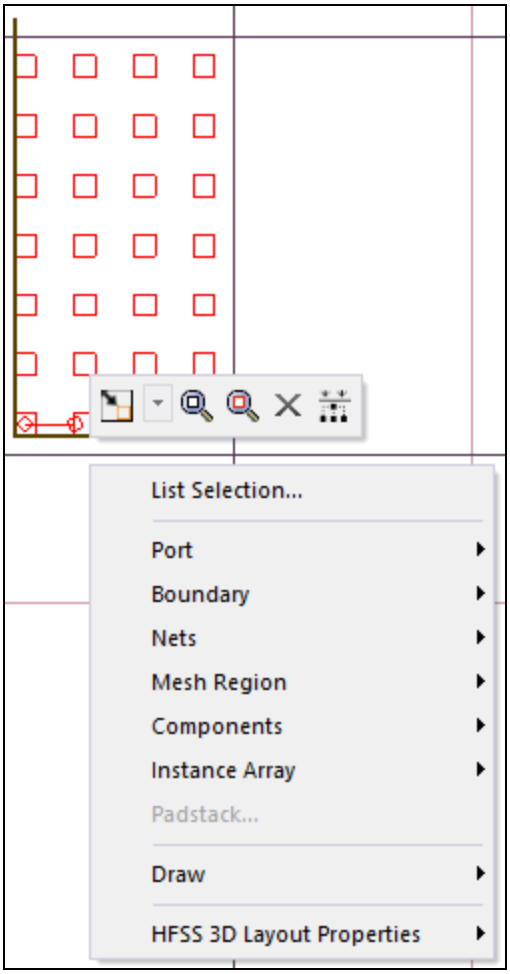
Instance arrays allow users to create arrays of cell instances to speed up the design process and reduce file size.

> **Note:**
>
> Instance arrays can only be created from a GDSII file. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform does not currently support editing a GDSII file in this context, but to import a GDSII file that includes instance arrays, refer to Importing GDSII Format Files in IC Mode.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

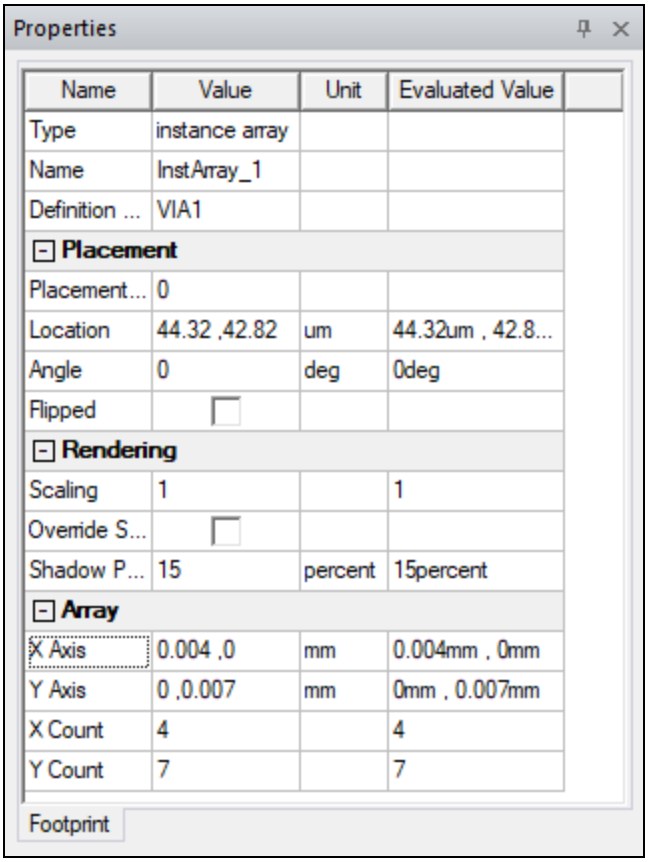
Selecting Instance Array

To select an instance array, click its axis, displayed as bold vertical and horizontal lines connecting in the bottom left-hand corner. Right-click on the array to view a contextual shortcut ribbon.

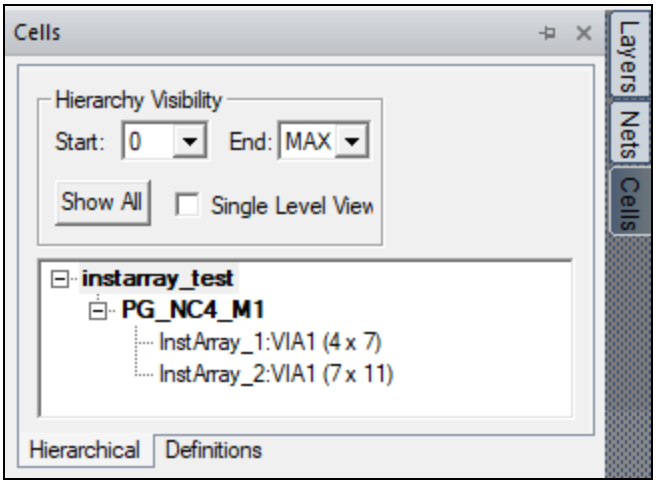


Selecting an array populates the **Properties** window. The array's editable parameters include **X Axis** and **Y Axis** (i.e., the values represent the array's offset from each axis), and **X Count** and **Y Count** (i.e., **X Count** is the number of cells present in the array in rows, while **Y Count** represents the number of cells present in the array in columns).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The array's hierarchical dimensions are visible in the **Cells** window.

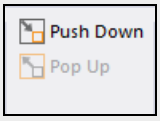


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

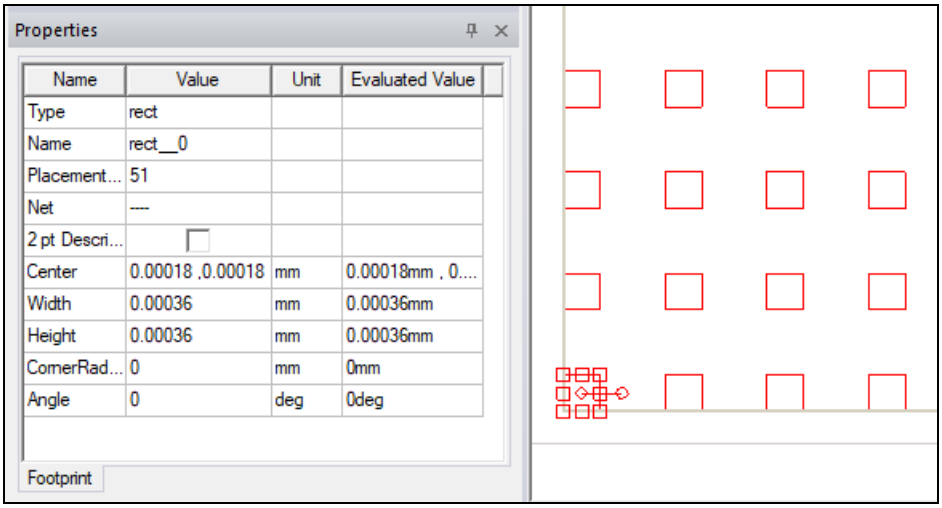
Navigating a Hierarchical Design

From the shortcut ribbon, select **Push Focus Down** and/or **Push Focus To** to navigate between components.

> **Note:**
>
> Double-clicking an instance array's axis will also focus on the cells (i.e., Push Down/Pop Up), as will clicking **Push Down**/**Pop Up** at the **View** ribbon.



Once the array's cells are highlighted, click the reference cell at the nexus of the vertical and horizontal axis lines to populate the **Properties** window with the cells' editable parameters.

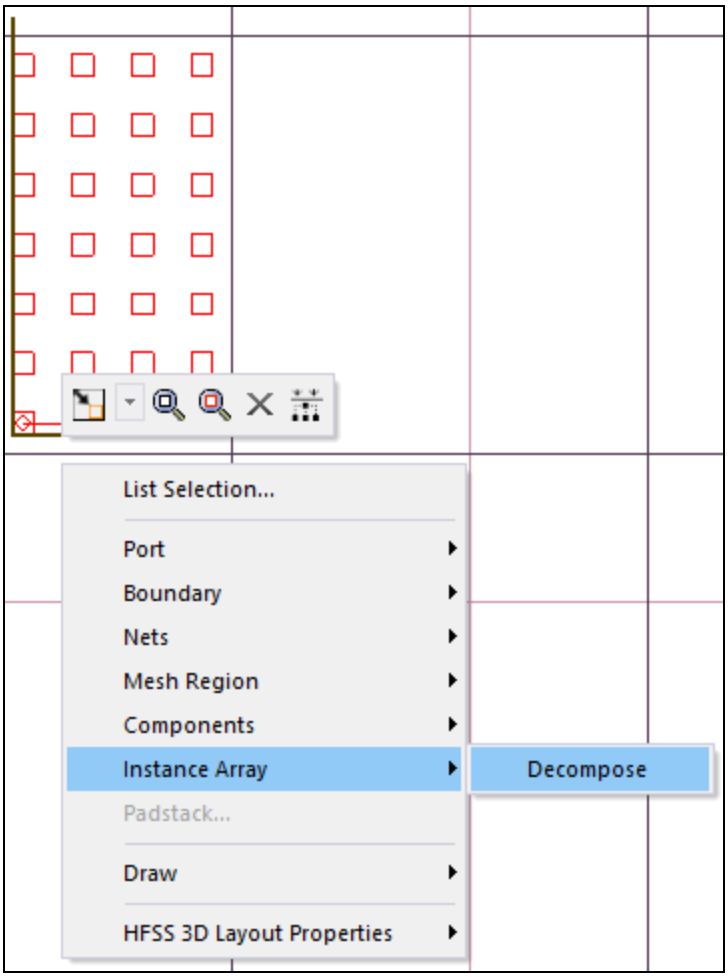


Dissolving an Instance Array

Dissolve an instance array to turn all of the array's cells into individual instances. To dissolve an array, do either of the following:

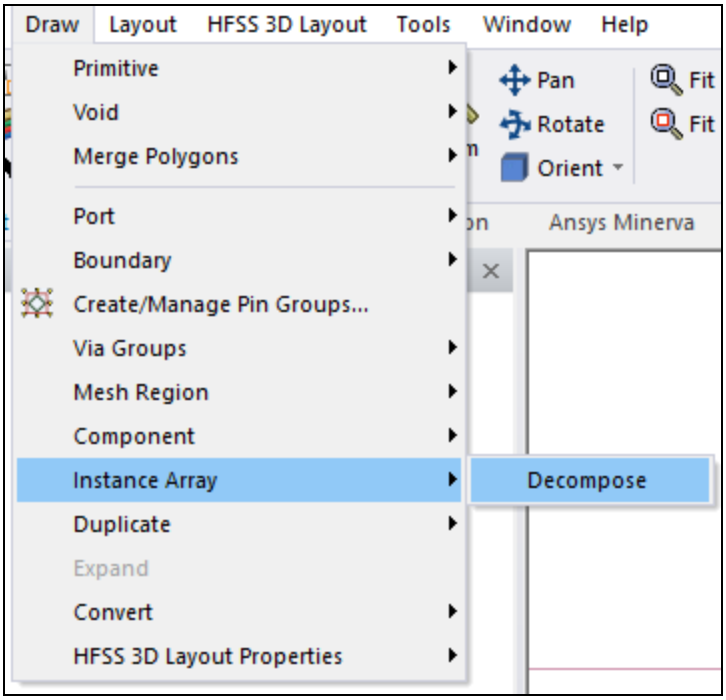
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click an appropriate instance array and select **Instance Array** > **Decompose**.



- With an instance array selected, navigate to **Draw** > **Instance Array** > **Decompose**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating Circuit Ports in IC Mode

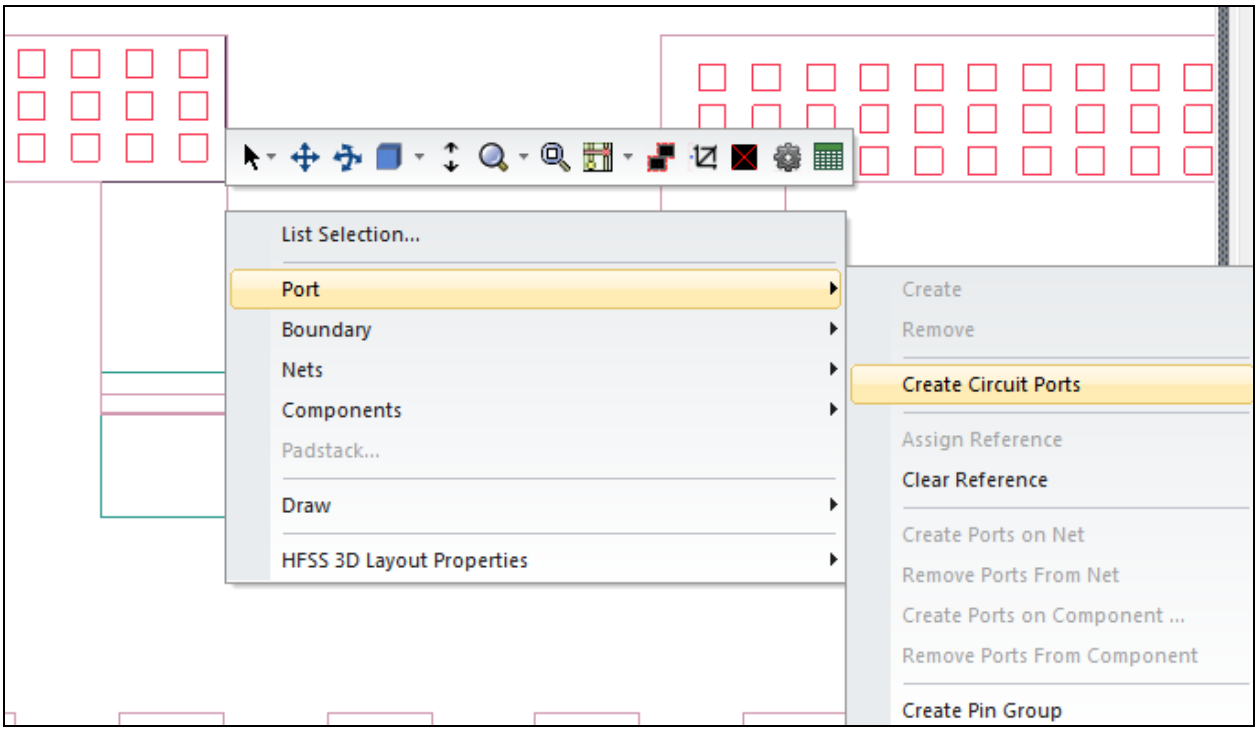
Circuit ports can be created the same way in both **General** and IC modes. Refer to the HFSS 3D Layout Creating a Circuit Port page for further information.

Creating a Point-to-Point Circuit Port

Follow these steps to create a circuit port.

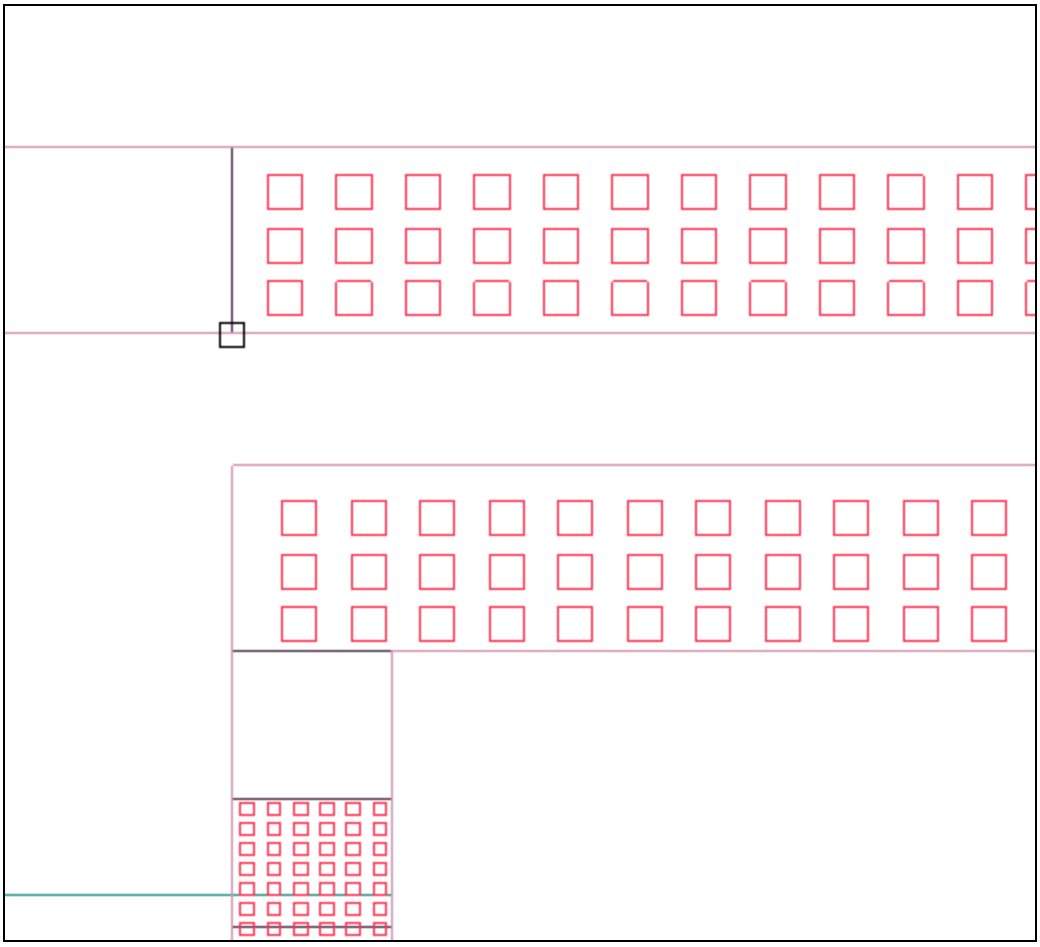
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click within the **Layout Editor** and select **Port** > **Create Circuit Ports**.



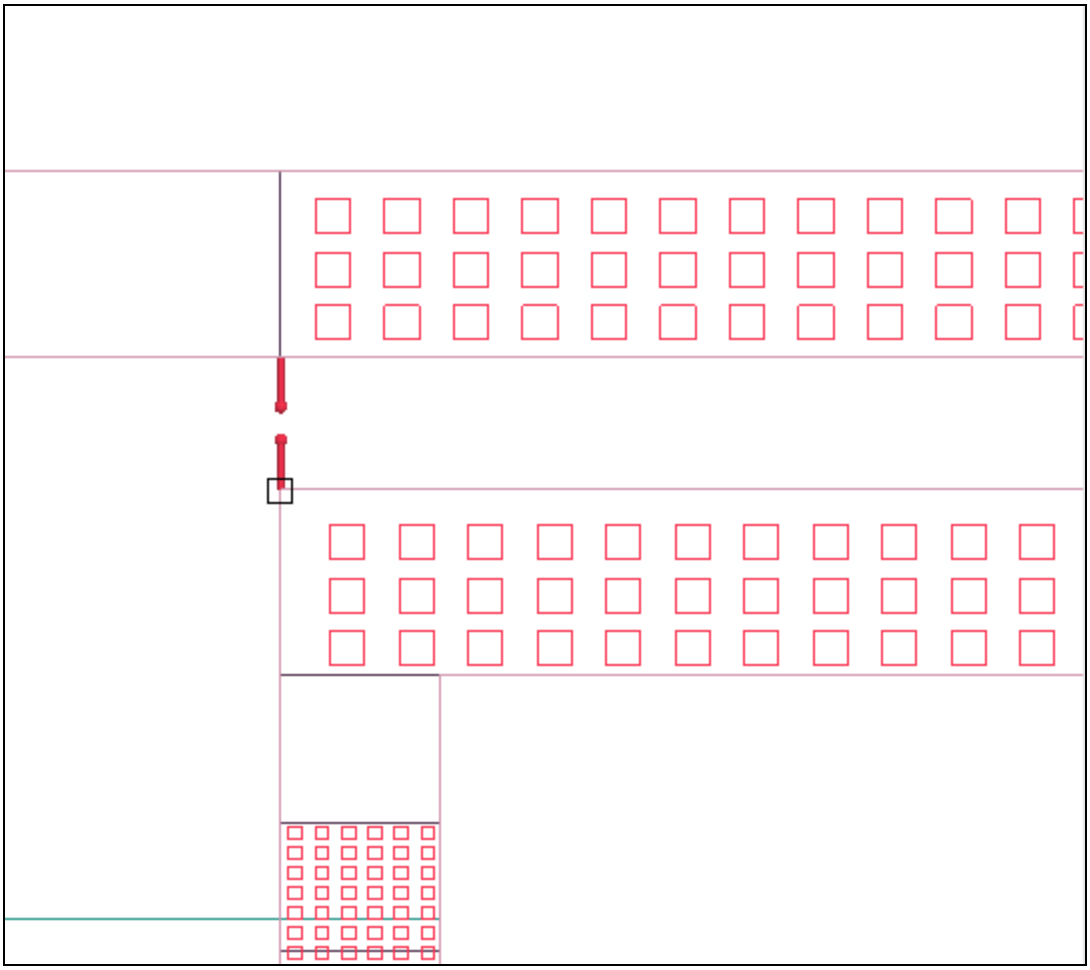
2. Hover the cursor over a conductor until the cursor changes shape (i.e., from an arrow into a square). Then left-click the conductor to select it as the positive terminal.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



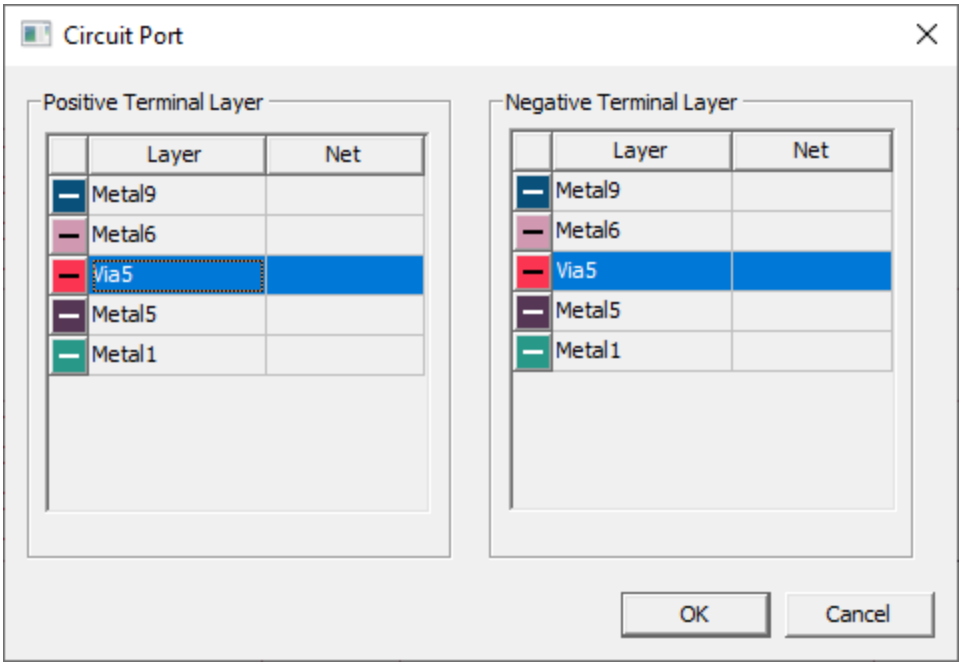
3. Repeat step 2, now dragging the port behind the cursor. Hover the cursor over a conductor until the cursor changes shape, then left-click the conductor to set it as the negative terminal.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Circuit Port** window will open.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



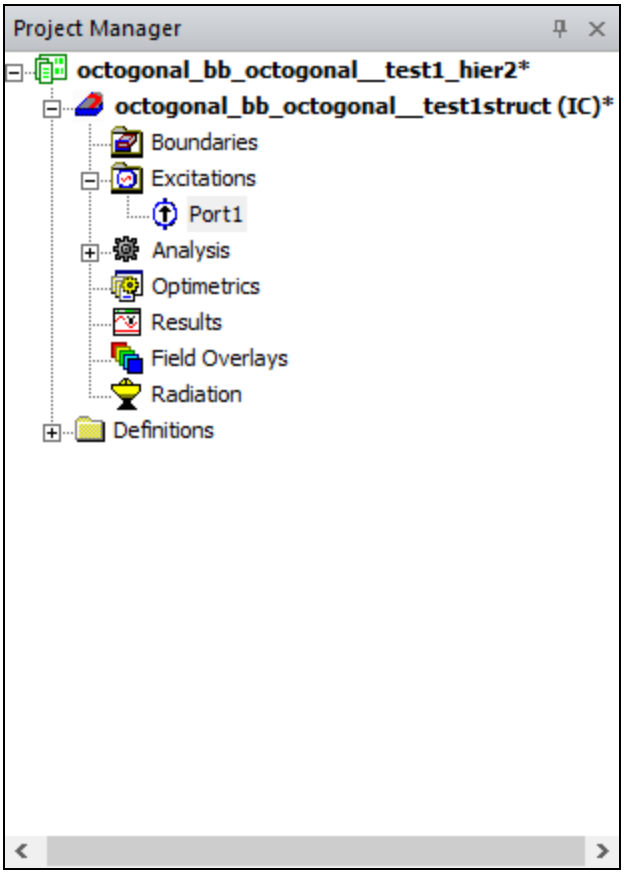
4. From the **Circuit Port** window, select a **Positive Terminal Layer** and a **Negative Terminal Layer**.
5. Click **OK** to close the **Circuit Port** window and finalize the new circuit port.

Editing Circuit Ports

Follow these steps to edit a circuit port's properties.

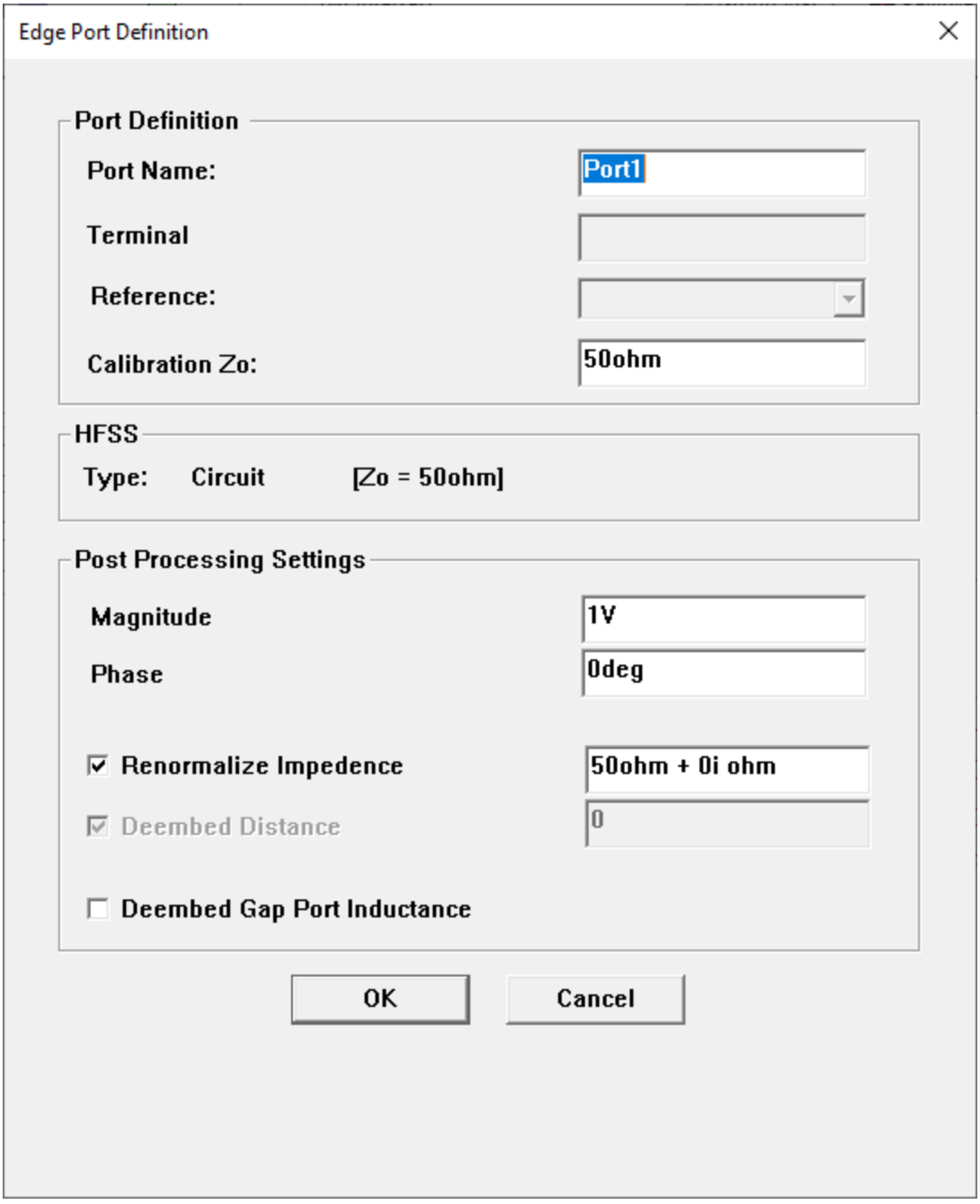
1. Navigate to the **Project Manager** window. Then expand the **Project Tree** > **[*Active Design Folder*]** > **Excitations** folder.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Double-click the selected port (e.g., **Port1**) to open the **Edge Port 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.



3. From the **Edge Port Definition** window, users can perform the following tasks:

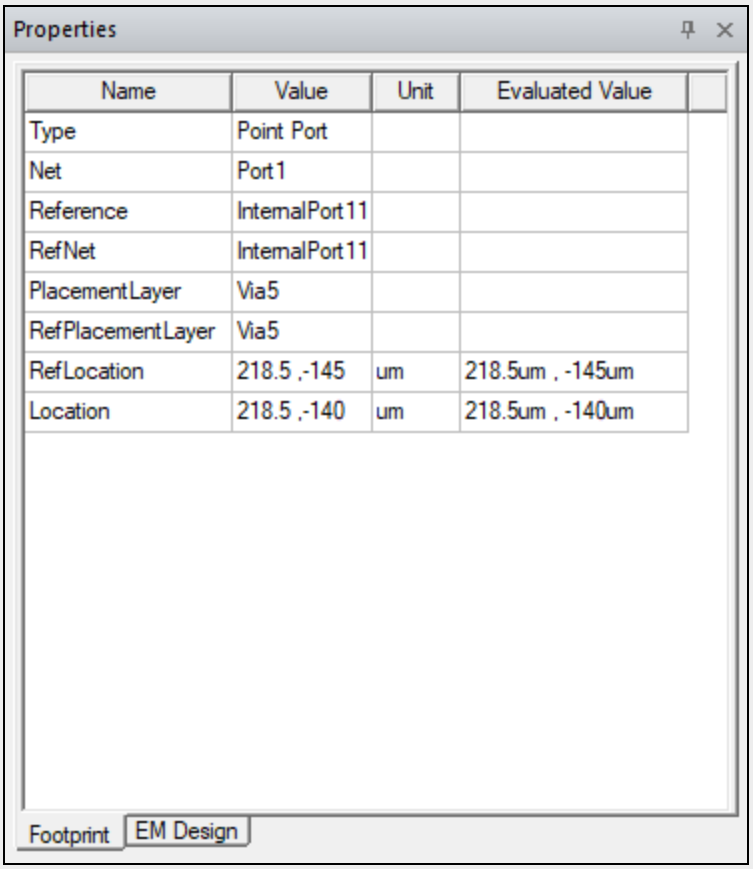
- Change the name of the pad port.
- Deembed the S-matrices.
- Vary the port excitation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Renormalize the port to a specific port impedance.
- Set up a reference to the nearest negative signal.

Note:

When a circuit port is selected, its parameters appear in the **Properties** window. Some values, such as the port's positive and negative terminal layers, can be changed directly in the **Properties** window.



3D Structures

A 3D structure is a special type of coordinate system (CS). Objects are grouped together as a 3D structure in the same way objects are grouped in a relative CS, with the following differences:

- When corresponding objects exist on two different layers of the project (e.g., a ground plane and a trace layer) and belong to the same 3D structure group, faces are automatically generated that span the various layers and connect the objects on the two

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

stackup layers resulting in 3D solid or planar geometry.

- Unlike in a CS, a 3D structure cannot be nested within another 3D structure. However, in addition to creating a 3D structure relative to the global CS, a 3D structure is created within any existing relative CS.

Activate a 3D structure to modify the objects within it (e.g., move, resize, et cetera). First, navigate to the **Layout Editor**. From **Draw**, select **3D Structure** and use the commands described in the following subsection.

Creating 3D Structures

When two or more objects on different layers are selected, the **Draw** > **3D Structure** > **Create** command connects the objects to create a single 2D or 3D object. Objects being connected must possess the same number of points.

- If the selected objects contain open lines, the created structure is a two-dimensional face (i.e., a cross-layer plate) that spans the different layers.
- If the selected objects contain closed lines, the created structure is a three-dimensional solid (i.e., a 3D Via) that spans the different layers. These 3D surfaces are included in the model. 3D structures also possess a material attribute.
- If the user selects **Create** and no objects are selected, the user must manually place the structure. The current CS is then set to the new 3D structure.

> **Note:**
>
> When **Draw** > **3D Structure** > **Create** is used to create a structure using a selection of closed polygon lines, the order in which the lines are selected is not significant. Lines are automatically ordered according to elevation before being connected by the closed surface. But when a structure is created using open lines, the selection of lines is order-significant (i.e., lines are connected in the order in which they are selected).

3D Ungroup

The following ungroup command is available when working with 3D structure objects.

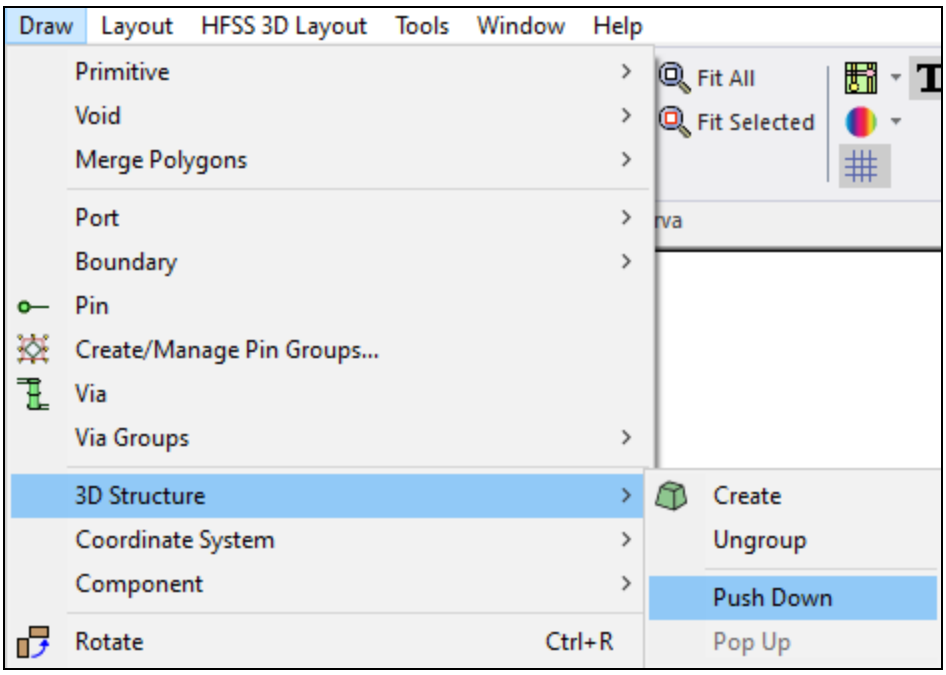
- The **Draw** > **3D Structure** > **Ungroup** command ungroups all selected objects and deletes the relative CS. Ungrouping 3D structures removes the connecting 3D surface between the objects that make up the structure without removing the objects.

3D Push Down and Pop Up

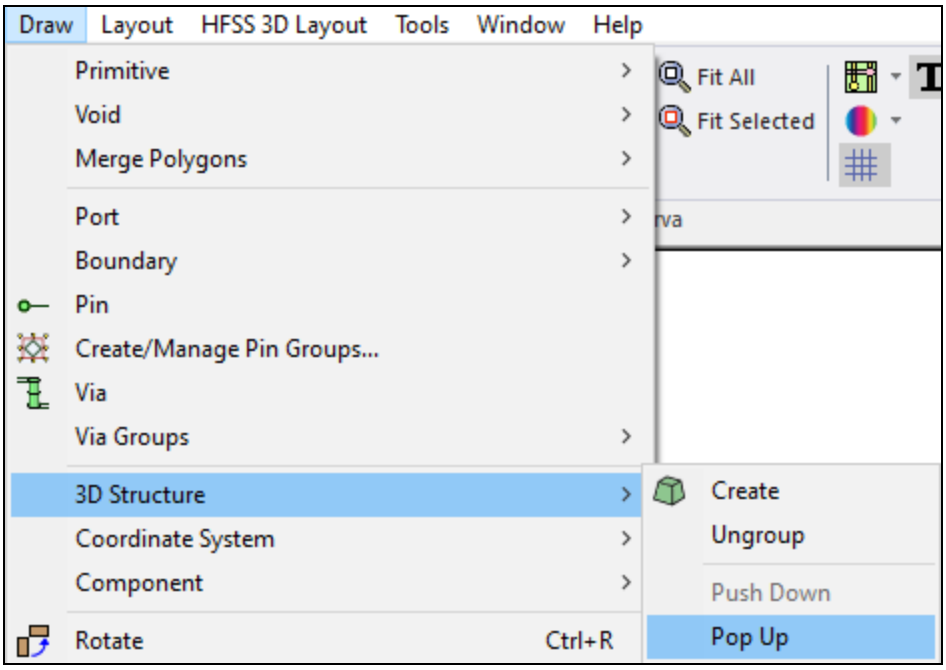
The 3D structure **Push Down** and **Pop Up** commands allow users to switch between a 3D structure and the global or relative CS on which it is based. Refer to the subsection within Coordinate Systems.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From **Draw**, select **3D Structure** > **Push Down** to activate the next lower level of the global CS. Repeat, as appropriate.



- From **Draw**, select > **3D Structure** > **Pop Up** to activate the next higher level of the global CS. Repeat, as appropriate.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hierarchical Design

Hierarchical design is a fundamental principle of HFSS 3D Layout that allows users to manage complex systems by breaking them down into manageable sub-components, such as single components or groups of component instances (i.e., components or MCAD 3D Components). In HFSS 3D Layout, the application of hierarchical design concepts is facilitated through the framework of Coordinate Systems (CS). By using coordinate systems, users can:

- Create individual components that can be designed, exported, and imported at different hierarchical levels within a relative or global CS, a modular approach streamlines the design process for complex systems.
- Parameterize individual components, or modify the parameters of an entire relative CS throughout the hierarchy.
- Create easily navigable schematic representations of large systems.
- Conduct simulations on individual components or an entire hierarchical design.

> **Note:**
>
> Hierarchical design can be used to define independent mesh domains (i.e., regions), which can then be meshed independently. This mesh fusion enables robust and rapid initial meshing of ECAD, MCAD 3D Components, and Component 2.0 representations. For an example of hierarchical design at work within the context of Mesh Fusion/Assembly for Hierarchical Design, refer to Mesh Fusion/Assembly for Hierarchical Design or Mesh Setup for Mesh Fusion/Assembly.

Refer to Component Placement in HFSS 3D Layout.

Coordinate Systems

From the **Layout Editor**, a CS forms the basis for organizing and managing objects (i.e., component instances) in a hierarchical design. A new coordinate system can include new object or group existent objects. If the CS is modified (e.g., offset or rotation angle), all grouped objects within are moved and rotate together. Each coordinate system is defined by an X-axis and a Y-axis. The origin (0,0) of each CS is located at the intersection of the axes. Each CS can be manipulated relative to the rest of the hierarchical design (i.e., users can specify a custom rotation angle of the X-axis of a CS relative to the parent CS's X-axis. The two types of coordinate systems in the **Layout Editor** are described as follows:

- The **Global** CS is the fixed, default CS for each new project. The global CS cannot be edited or deleted.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A **Relative** CS is dynamic within the global CS. A relative coordinate system's origin and orientation are relative to any existing objects or to the origin and orientation of its parent CS. All relative coordinate systems can be separate, albeit nested with the global CS, or they can be in a multi-level nested arrangement. Modifying a relative CS affects all of the objects within CS and any nested coordinate systems.

Note:

- Switch between global and relative coordinate systems by changing the *active CS*. The colors of objects outside the active CS are lightened to differentiate the appearance of active CS objects.
- Activate a CS to select and modify the objects within (e.g., move, resize, et cetera).
- User-defined coordinate systems are saved with the active project.

Creating a Coordinate System

From **Draw**, select **Coordinate System** > **Create**. The **Create** command allows users to define a relative CS whose origin and orientation can be set relative to an existing object or set relative to another existing CS to create nested coordinate systems. When a relative CS is active, and a new CS is created, the new CS becomes a child of the CS on which it was based. Use **Push Down** and **Pop Up** commands to navigate the nested CS hierarchy.

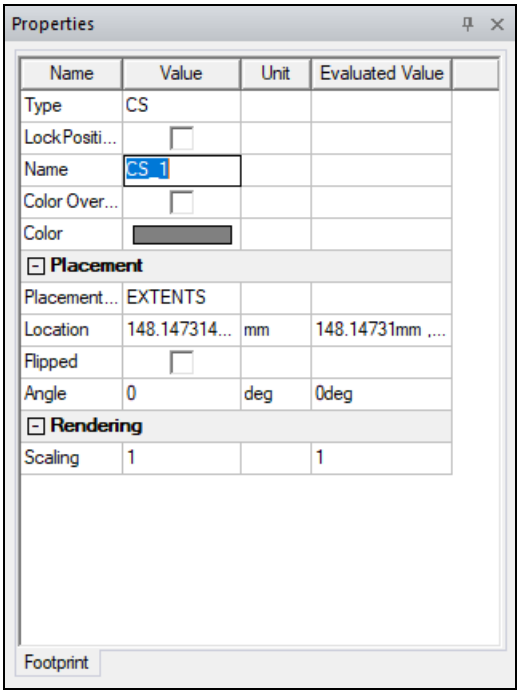
New Coordinate System with No Object Selected

If no edge, port, or port instance is selected, the command creates a new relative CS. The cursor becomes a circle with XY crosshairs to indicate a new CS is being created. When positioned over a grid snapping point, the cursor becomes a diamond. Click in the **Layout Editor** to position the new relative CS. Objects created subsequently are grouped together and positioned relative to the origin of the new CS. This relative placement is indicated by each object's positional values in the **Properties** window.

New Coordinate System with Edge, Port, or Port Instance Selected

If an edge, port, or port instance **is** selected, the command creates a new CS relative to the selected object. By default, the origin of the new CS is located at the midpoint of the edge, port, or port instance, whether straight or curved. The new CS axes are rotated to align with a selected object that is not parallel to the global coordinate axes. Adjust the position of the origin along the length of a selected object by altering its **Edge Position** value in the **Properties** window. Specify a value between **0** and **1** (default is **0.5**). Objects created subsequently are grouped together and positioned relative to the origin and rotation angle of the new CS. If applicable, enter a custom **Name** for the CS in the adjacent 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 Coordinate System with a Non-Applicable Object Selected

If a non-applicable object is selected, the selection is ignored and cleared.

Selecting a Coordinate System

The CS **Edit** command allows you to switch the active CS between multiple existing coordinate systems (i.e., the global CS and all relative coordinate systems). Follow these steps to select and edit a CS.

1. Select **Draw** > **Coordinate System** > **Edit** to open the **Select Coordinate System** window.
2. Select a CS to edit. The selected CS is temporarily activated in the layout to show which objects are affected.
3. After making any appropriate edits, click **Select**, and the modified CS becomes active in the **Layout Editor**.

Push Down and Pop Up Commands

The **Push Down** and **Pop Up** commands are multipurpose, even within the context of coordinate systems. They are used to navigate between levels of the CS hierarchy, push into components, and navigate the context of sub-designs. If a CS is selected, the **Push Down** and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pop Up commands refer to the nested coordinate systems within. Using **Push Down** and **Pop Up**, users can navigate between different contexts in the design hierarchy in HFSS 3D Layout. They allow you to modify the footprints of components (e.g., coordinate systems, discrete devices, et cetera.) by pushing the components down and changing their pad position.

> **Note:**
>
> For sub-circuits and components, the **Push Down** and **Pop Up** commands are accessible by right-clicking in the **Layout Editor** or via the **Layout** menu. There are also special variants for 3D structures. For information about selecting the active 3D structure, refer to 3D Structures.

If neither the **Push Down** nor **Pop Up** command is available, the global CS or hierarchical design is active. You cannot rise higher than the global CS or push down unless a relative CS or sub-circuit is selected. When a relative CS is created for the first time, it is created as a child CS relative to the global CS, similar to how a sub-circuit is related to the hierarchical design in which it is added.

For regular user-defined coordinate systems, the **Push Down** and **Pop Up** commands are located at **Draw** > **Coordinate System** .

The **Push Down** and **Pop Up** commands allow you to switch between different nested CS levels or from a relative CS to the global CS.

- **Draw** > **Coordinate System** > **Push Down** activates the selected CS from its parent global or relative CS.
- **Draw** > **Coordinate System** > **Pop Up** switches on the currently active CS to its parent (up one level). If a single-level relative CS or the highest level of a set of nested CSs is active, popping up activates the global CS.

Users can also push down within a relative CS or design by double-clicking the rectangular border around its objects. The border does not display if component visibility is deactivated for the layers with objects comprising the CS or sub-design. Like **Push Down**, double-clicking only moves down one level at a time. You cannot pop up to a higher-level CS or sub-design by double-clicking. You can only switch to a child of the current CS or sub-design (e.g., assume CS_1 and CS_2 are not nested, but both are defined relative to the global CS. If CS_1 is active, you cannot activate CS_2 by double-clicking it. First, **Pop Up** to activate the global CS. Then, double-click CS_2.) .

The following group/ungroup commands are available when working with objects in a relative CS.

- **Draw** > **Coordinate System** > **Group Into CS** groups all selected objects into a new relative CS.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Draw** > **Coordinate System** > **Ungroup** ungroups all selected objects and deletes the relative CS.

To group or ungroup a 3D structure, refer to 3D Structures.

DC IR and Icepak Simulation on Hierarchical Designs

DC IR and Icepak simulations support hierarchical designs, enabling accurate simulation of electrical and thermal behavior across stacked components such as packages, boards, and nested sub-designs. These solvers operate on unified layouts that may include package-on-PCB assemblies or package-on-package stacks exported from SIwave.

> **Note:**
>
> This feature is currently in beta. Beta features must be activated prior to use. Refer to Activating the ECAD-ECAD Hierarchical Design Beta Feature for instructions. Refer to Hierarchical Design.

If the design originates in SIwave, it can be exported to Electronics Desktop for full modeling of placement and connectivity. Once imported, simulations can be launched directly on the hierarchical structure.

1. Start with a hierarchical layout in SIwave or Electronics Desktop.
2. Export the layout to Electronics Desktop using Export HFSS 3D Layout Project or Icepak Export, if starting from SIwave.
3. Run DC IR or Icepak simulations, which support hierarchical design.

DC IR computes voltage drop and current flow across the full structure, while Icepak models temperature distribution based on power dissipation. These solvers can be coupled iteratively to refine results using temperature-dependent material properties. Together, they provide a comprehensive solution for analyzing power integrity and thermal performance in complex assemblies.

SIwave Export to Electronics Desktop

SIwave cannot simulate coupling between non-coupled stacked packages and the board. Exporting these configurations from SIwave to Electronics Desktop enables full modeling of hierarchical assemblies that include physical placement but lack direct coupling in SIwave. In SIwave, non-interacting stackups are used to represent stacked configurations without simulating the electrical interaction between them. When exported to Electronics Desktop, these configurations are instantiated as a true hierarchical design in which all components are physically positioned and electrically connected. This allows DC IR and Icepak simulations to operate on the complete structure, capturing power and thermal behavior across the full stackup.

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, use the scripting command **`oDoc.ScrExportAEDT()`** to export the design. All hierarchy levels will be populated, and packages mounted in multiple locations will be represented as single designs with multiple instances.

Note:

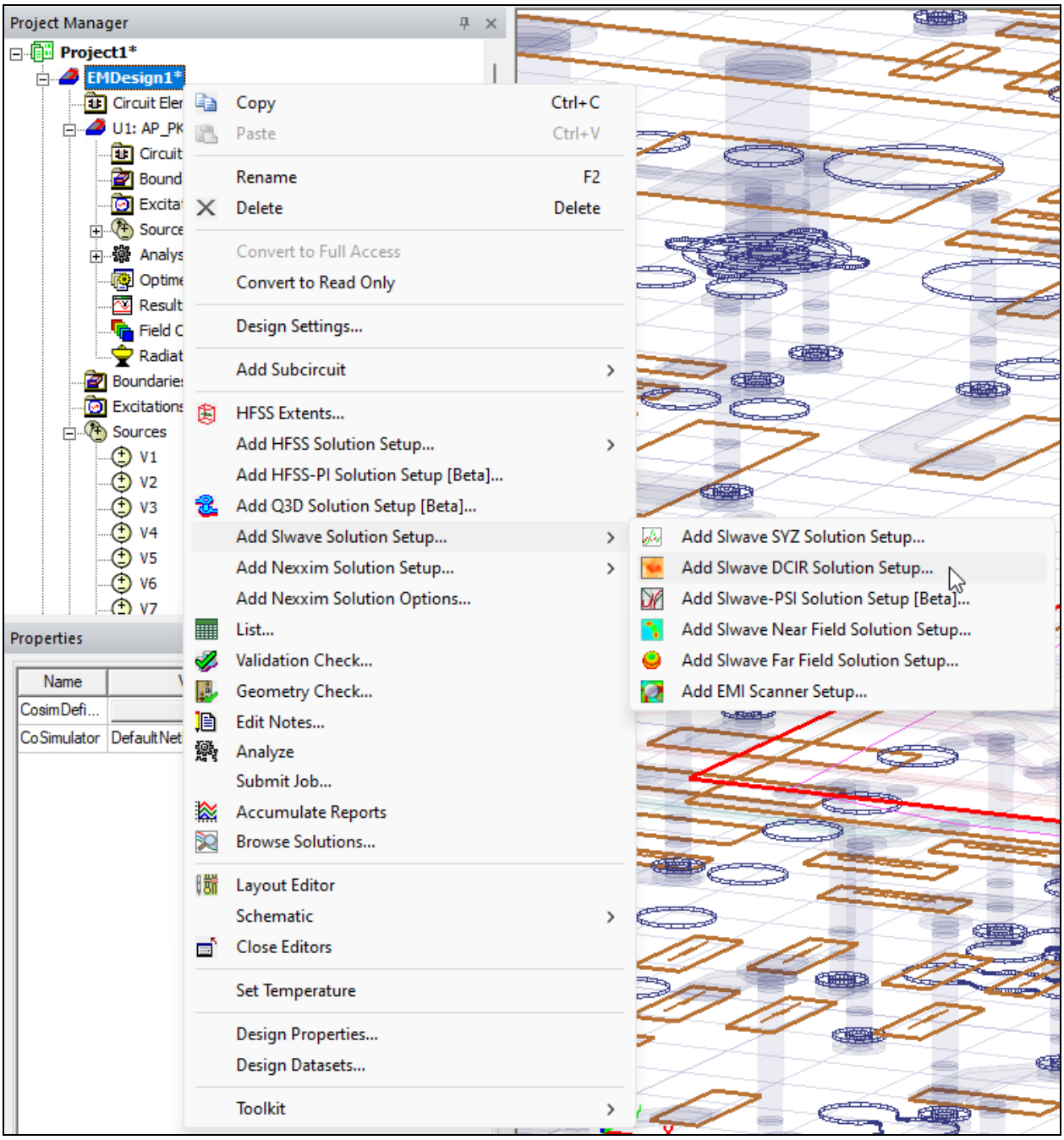
Icepak simulations impose a configuration constraint: each cell in the hierarchy must be instantiated only once. To instantiate multiple copies, the cell must be duplicated so that each instance originates from a unique source. This restriction does not apply to DC IR. When exporting from SIwave using the Icepak export option, this duplication is handled automatically. Other EDB-based exports may allow multiple instances and may require manual adjustment for Icepak compatibility.

DC IR Simulation on Hierarchical Designs

DC IR simulation on hierarchical layouts enables accurate modeling of voltage drop and current flow across complex assemblies that include multiple physically stacked sub-designs. In this workflow, users define voltage and current sources across packages, boards, and nested components, allowing the solver to compute power distribution throughout the entire structure. The simulation accounts for the electrical connectivity between sub-designs, including interposers, solder balls, and shared nets, and produces a unified solution that reflects the full electrical behavior of the system. This approach supports advanced power integrity analysis in multi-component layouts without requiring flattening or manual merging 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.

1. Open a hierarchical layout in Electronics Desktop (e.g., package on PCB).
2. Run a DC IR simulation on the top-level design.

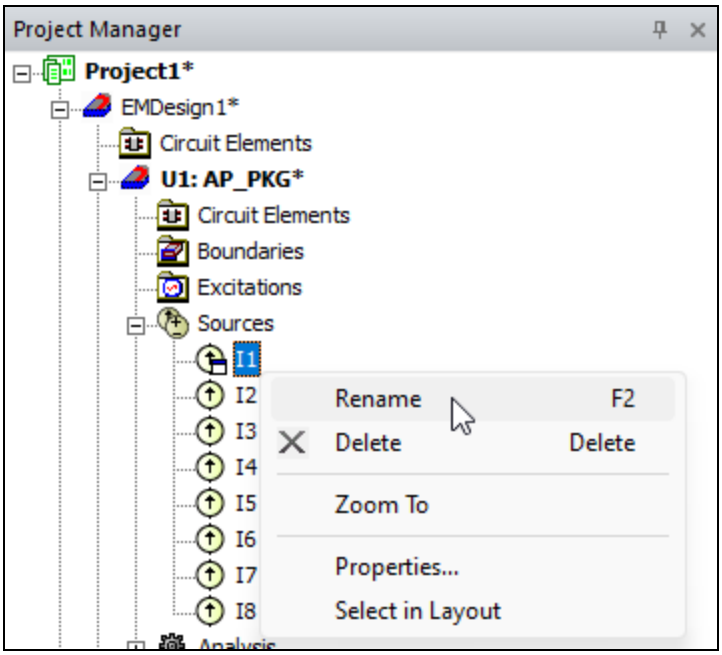


3. Define current and voltage sources:
   - Voltage sources define where voltage is applied (e.g., power rails).
   - Current sources define where current is drawn (e.g., IC pins).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

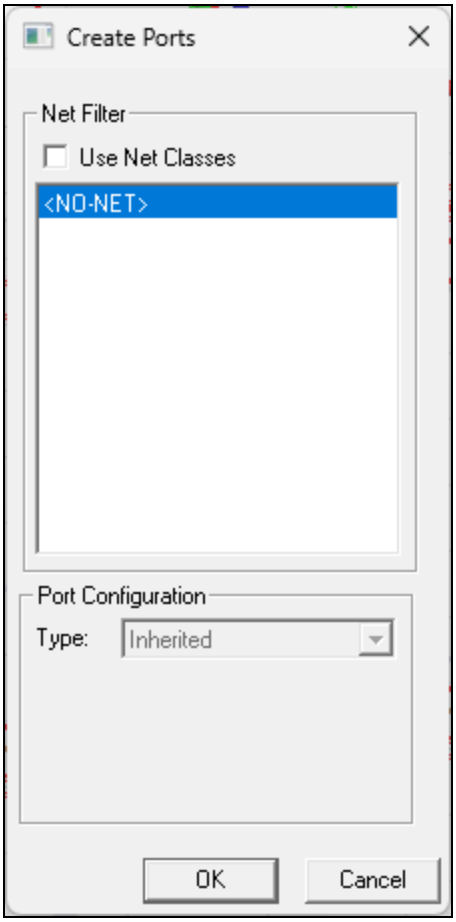
> **Note:**
>
> The software allows duplicate names across sub-designs, but the simulation will fail due to naming collisions.

4. Ensure all sources across all sub-designs have **unique names** by doing either of the following:
    - From the **Project Manager** window, expand the ***Project*** > *active design* > each *sub-design* > and **Sources**. Then right-click each source, select **Rename**, and enter a new name for the source.



    - Promote sources from sub-designs to the top-level design. During promotion, sources with duplicate names will be renamed:
        a. From within the **Layout Editor**, right-click the sub-design and select **DCIR** > **Create Current Sources on Components** to open the **Create Ports** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. From the **Create Ports** window, do the following:

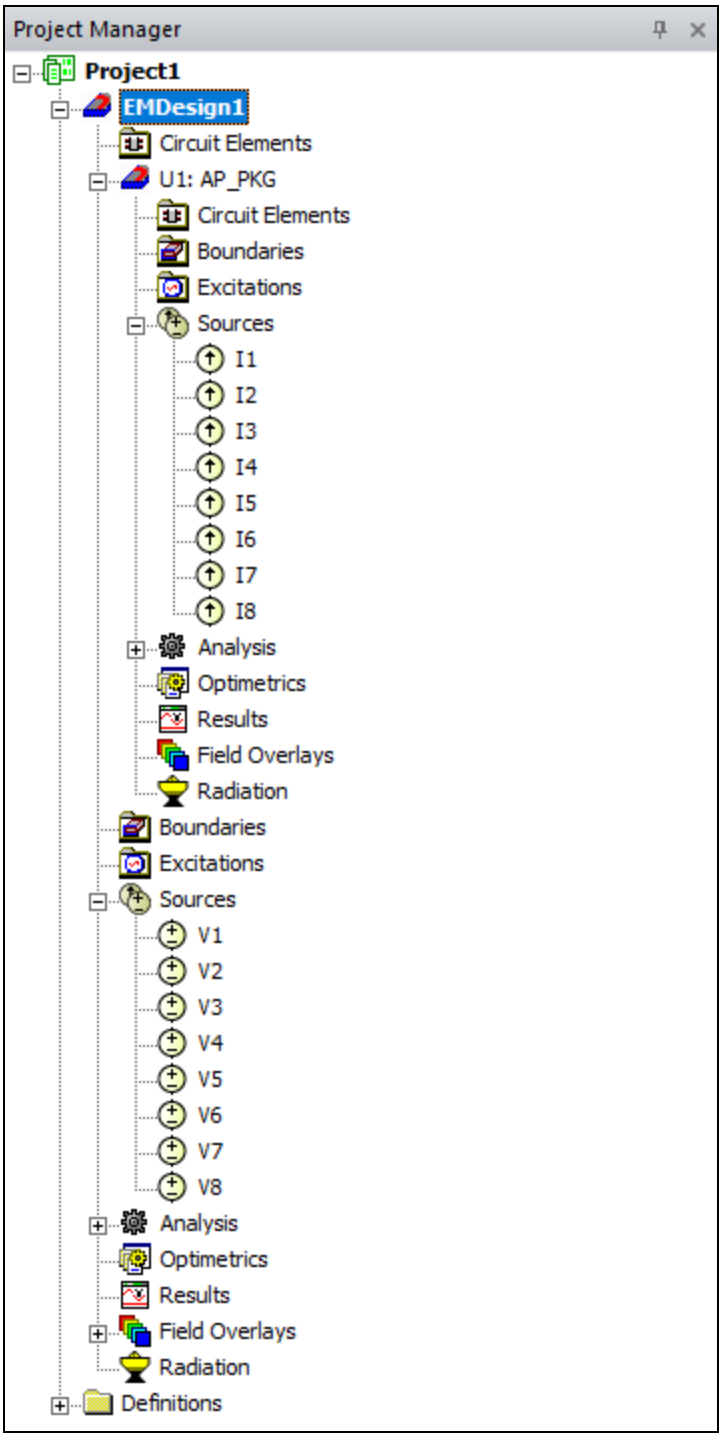
- From the **Net Filter** area, check the **Use Net Classes** box, if applicable.
- Select a net from the list.
- From the **Port Configuration** area, select a **Type** of port from the drop-down menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click **OK** to promote the connected sources to the top-level design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

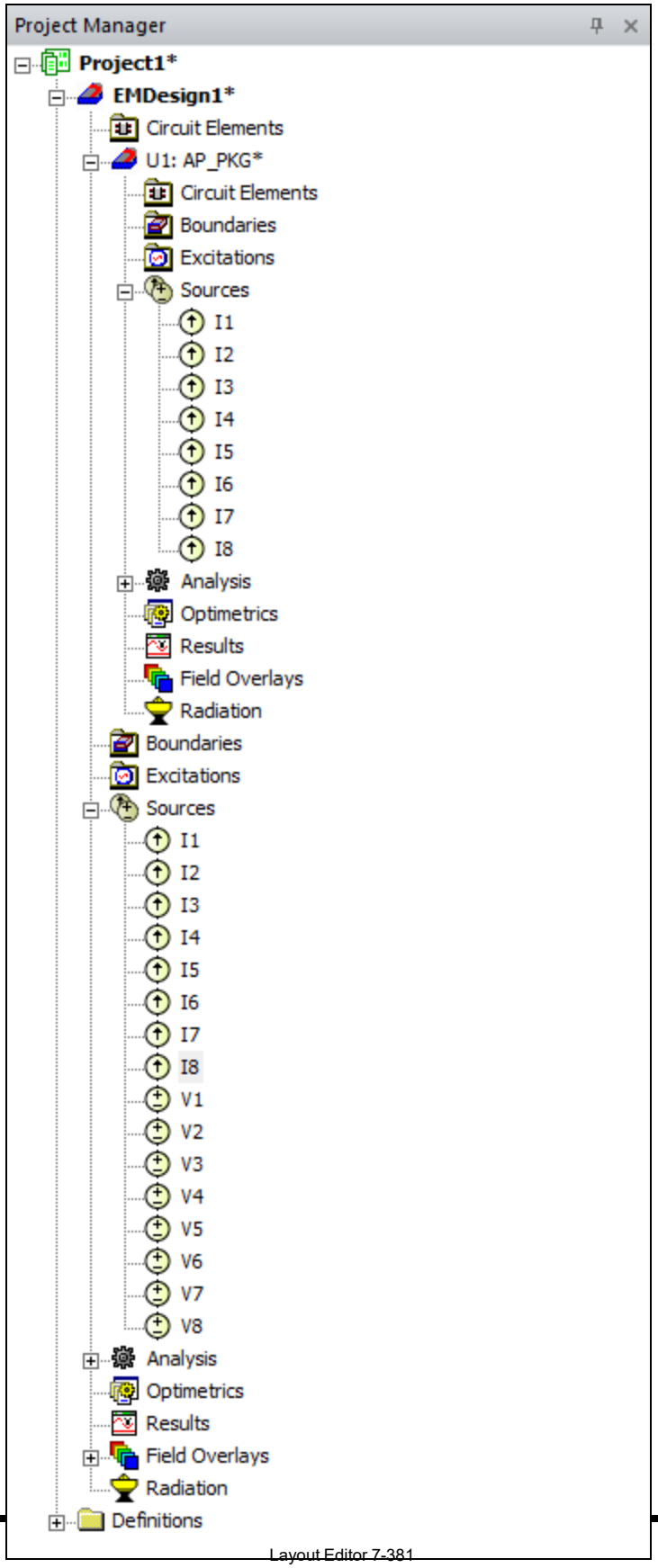
Before – Sources are spread across two levels of the hierarchical design.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After – Sources are shown on the top level of the hierarchical 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.

Note:

Source promotion is strongly recommended for all DC IR and Icepak simulations. It ensures name uniqueness across the hierarchy and prevents simulation failures due to naming collisions. This is especially important when a cell is instantiated multiple times, as each instance would otherwise create duplicate source names. In cases where all source names are already unique and no cells are reused, promotion is not strictly required, but it remains a best practice.

Two-Way Coupled Icepak Simulation

Two-way coupled simulation between DC IR and Icepak enables thermal-electrical feedback modeling across hierarchical designs. In this workflow, the electrical power distribution computed by DC IR is used as the heat source input for Icepak, which in turn calculates the resulting temperature distribution. Because electrical conductivity is temperature-dependent, the updated temperature profile is then used to refine the material properties in the DC IR model. This feedback loop continues for several iterations, allowing both solvers to converge on a solution that reflects the mutual influence of electrical and thermal behavior.

1. Set a uniform temperature and enable feedback in the HFSS 3D Layout design.
2. Run a DC IR simulation to compute power distribution.
3. Add an Icepak design to the active project and insert a PCB-native component.
4. Add two-way coupling to the Icepak design.
5. Run an Icepak simulation to compute temperature distribution.
6. Repeat DC IR and Icepak solves for 3–4 cycles until convergence.

This iterative process ensures accurate modeling of temperature-dependent electrical behavior. Refer to Coupling SIwave DC IR Loss. It is fully supported on hierarchical designs, including those with multiple packages, nested components, and virtually mounted structures.

Coordinate Systems and Hierarchy Navigation

Coordinate systems provide the structural framework for organizing components in hierarchical designs. Each object or sub-design is positioned within either a global or relative coordinate system, which defines its origin, orientation, and nesting behavior. The global coordinate system serves as the fixed reference for the entire layout, while relative coordinate systems allow localized control over placement and rotation. Users can navigate between hierarchical levels using **Push Down** and **Pop Up** commands, which activate different coordinate contexts and expose the contents of nested designs. Grouping objects into a relative coordinate system enables modular editing and reuse, while ungrouping dissolves the local context and returns objects to the parent layout. These tools streamline layout management and support precise

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

control over component positioning, especially in multi-level assemblies. Refer to Coordinate Systems.

- The **Global** coordinate system is fixed and cannot be edited.
- **Relative** coordinate systems are defined per component or sub-design and can be nested.
- Use **Push Down** and **Pop Up** to navigate between hierarchical levels.
- Use **Group Into CS** and **Ungroup** to manage component placement.

> **Note:**
>
> Coordinate system hierarchy affects layout navigation and component grouping, but does not directly impact DC IR or Icepak simulation fidelity unless sources or components are hidden or misaligned.

Validation and Visualization

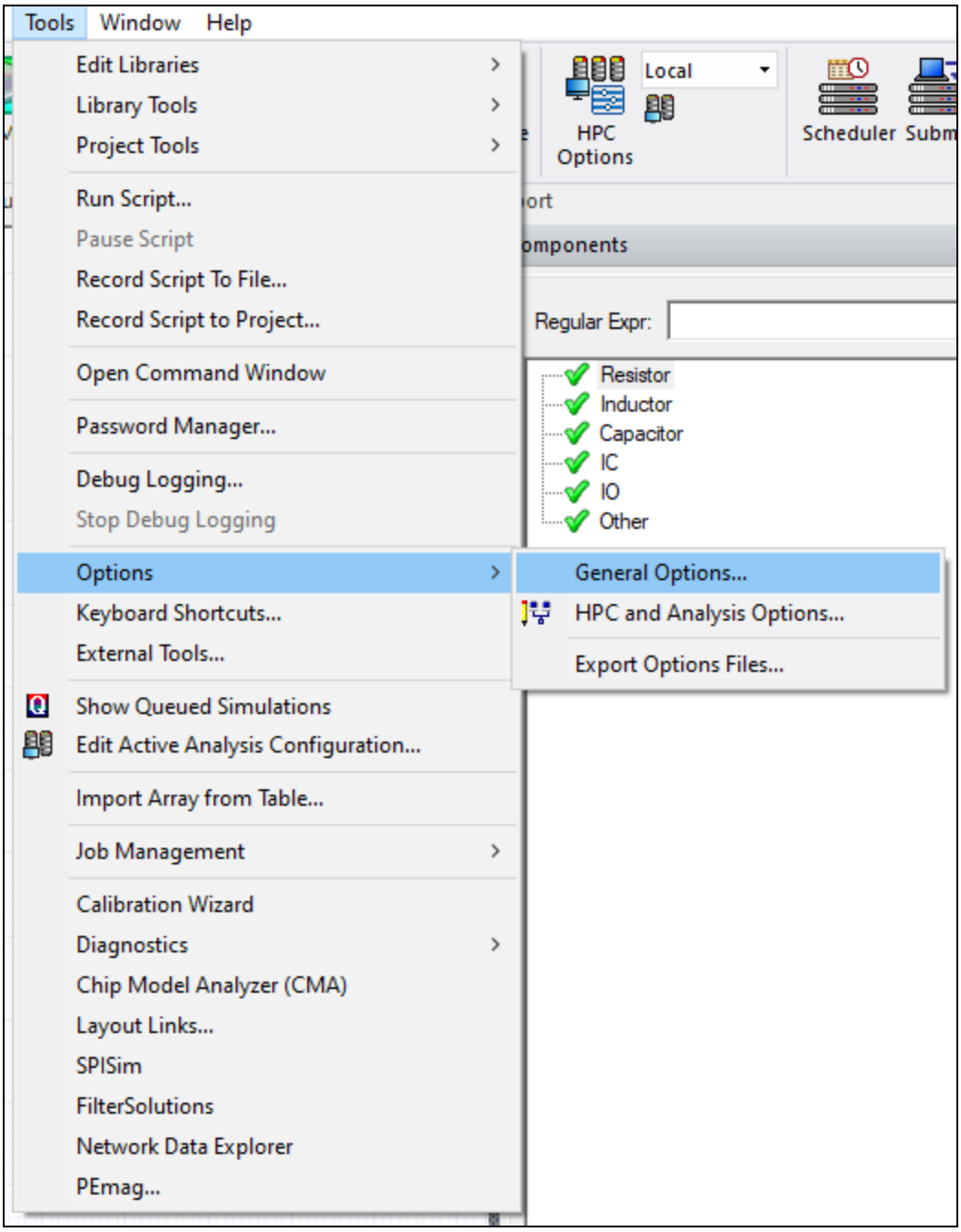
Simulation results can be validated using field overlays and post-processing tools in Electronics Desktop. The **Create Field Plot** window supports visualization of layers from linked hierarchical designs. The **Layout Editor** presents the full design with stacked packages, and the **Project Manager** window displays nested hierarchy with multiple instances of the same linked design. Refer to Postprocessing and Generating Reports.

Activating the ECAD-ECAD Hierarchical Design Beta Feature

Follow these steps to enable the ECAD-ECAD hierarchical design beta feature.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

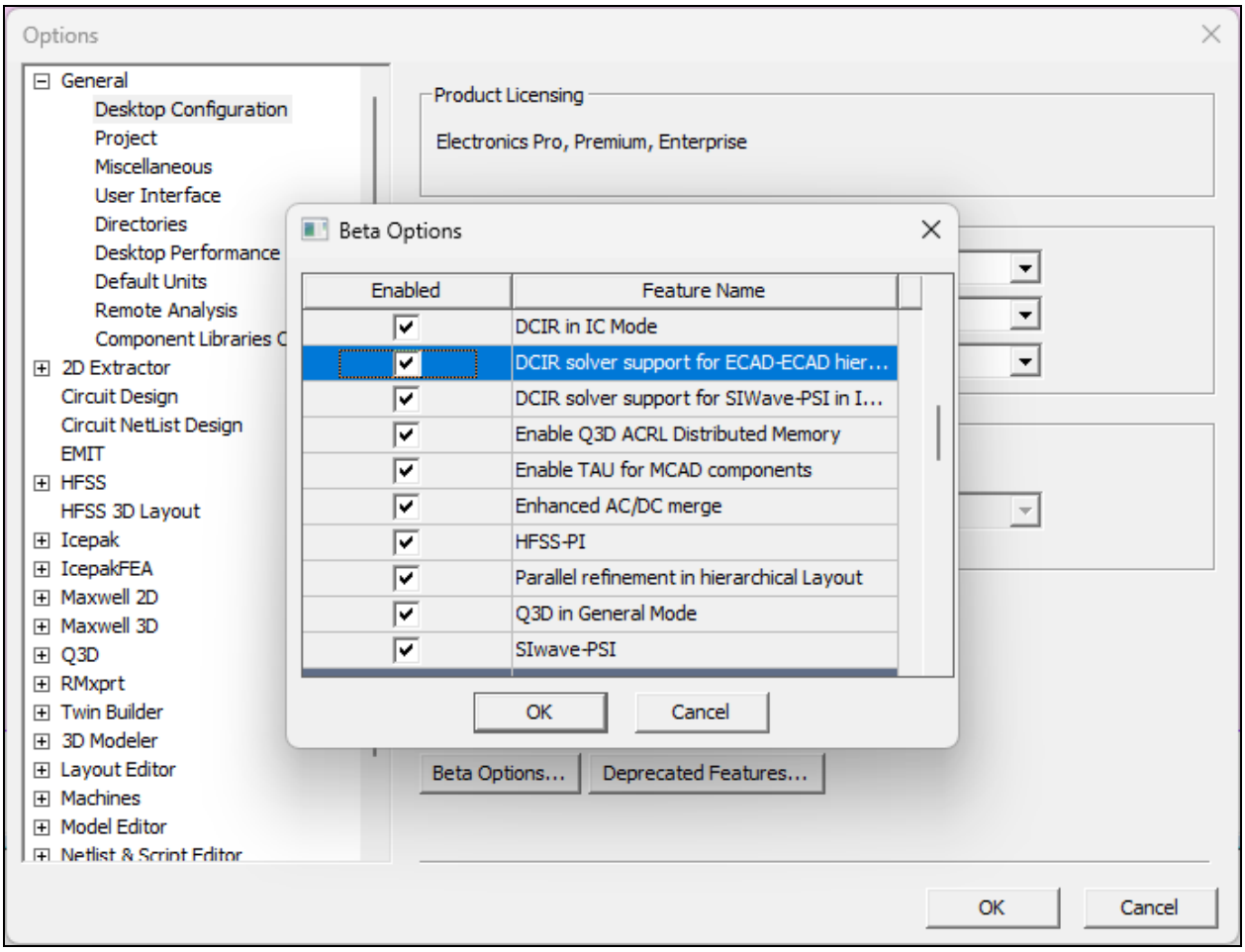
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window



2. Expand **General** > **Desktop Configuration** and click **Beta Options** to open the **Beta Options** window

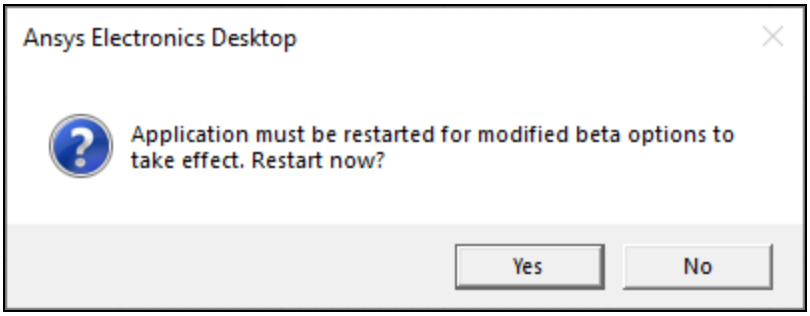
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scroll down to the **HFSS 3D Layout** section and check the box adjacent to **DC IR solver support for ECAD-ECAD hierarchy**.



4. Click **OK** to close the **Beta Options** window.
5. Click **OK** to close the **Options** window and open a prompt to restart the software. Users may be asked to save their work.

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 **Yes** to restart immediately.
7. After restarting the software, simulation of hierarchical design will now function as described in the previous section.

The procedure is complete.

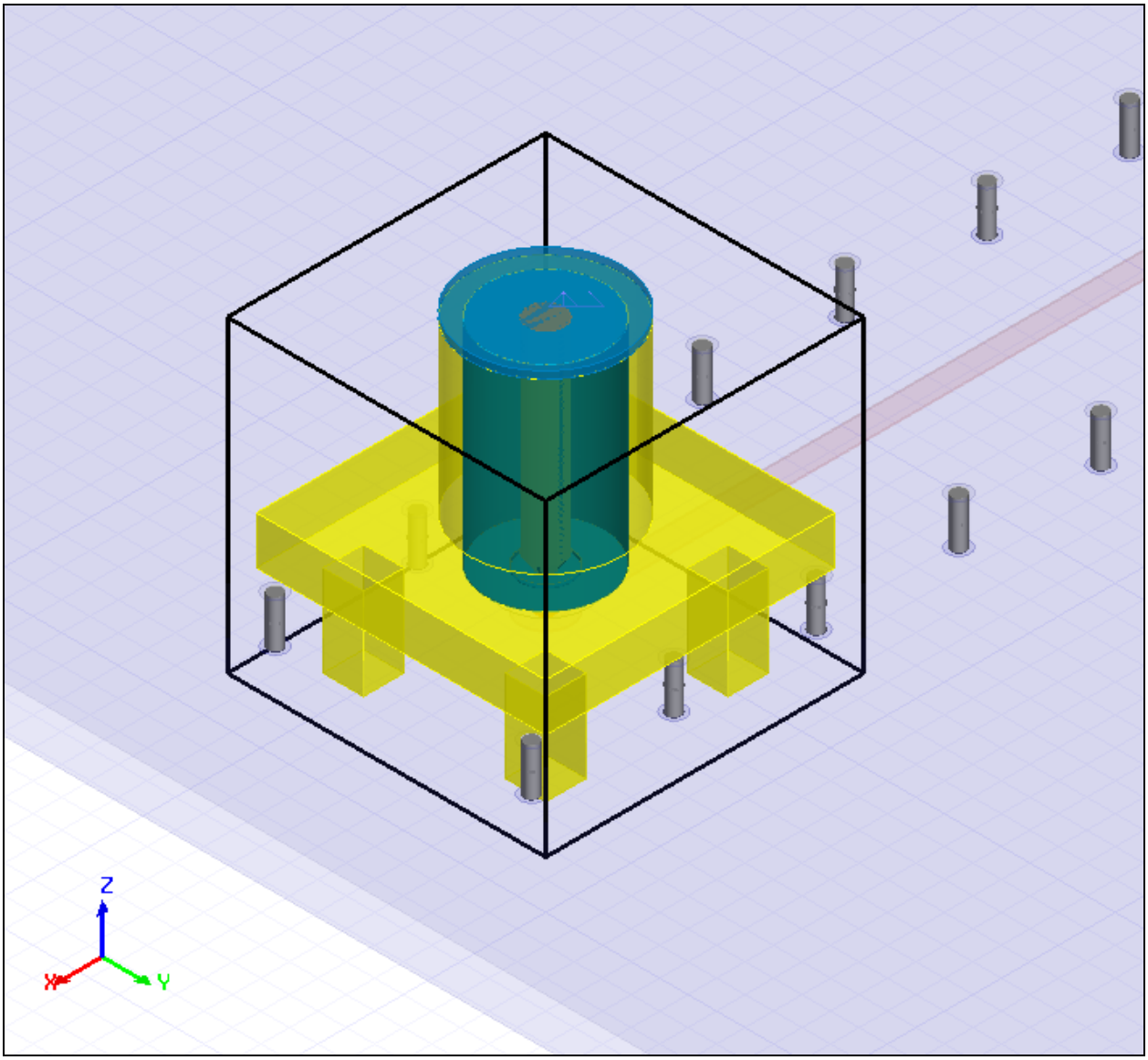
Mesh Fusion/Assembly for Hierarchical Design

In HFSS 3D Layout, a hierarchical design is defined by the top-level layout and a collection of component instances. Component instances can be of different types, most commonly other components, 3D Components (MCAD), and Component 2.0 representations. For mesh assembly/fusion (Mesh Fusion), this hierarchy is used to define independent mesh domains, with then *n*top-level layout referred to as the “native” region. In allowing separate regions to be meshed independently, Mesh Fusion enables robust and rapid initial meshing.

> **Note:**
>
> To use Mesh Fusion, the native region must encapsulate all component instances. For a component instance to be modeled as a separate mesh domain, check the **Mesh Fusion** box in the "Access the Mesh Settings Window" on page 7-388 window.

By default, components selected for Mesh Fusion are assigned a bounding-box envelope for its mesh region. If a 3D component has explicitly defined “mesh regions”, those are also available for envelope assignment.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



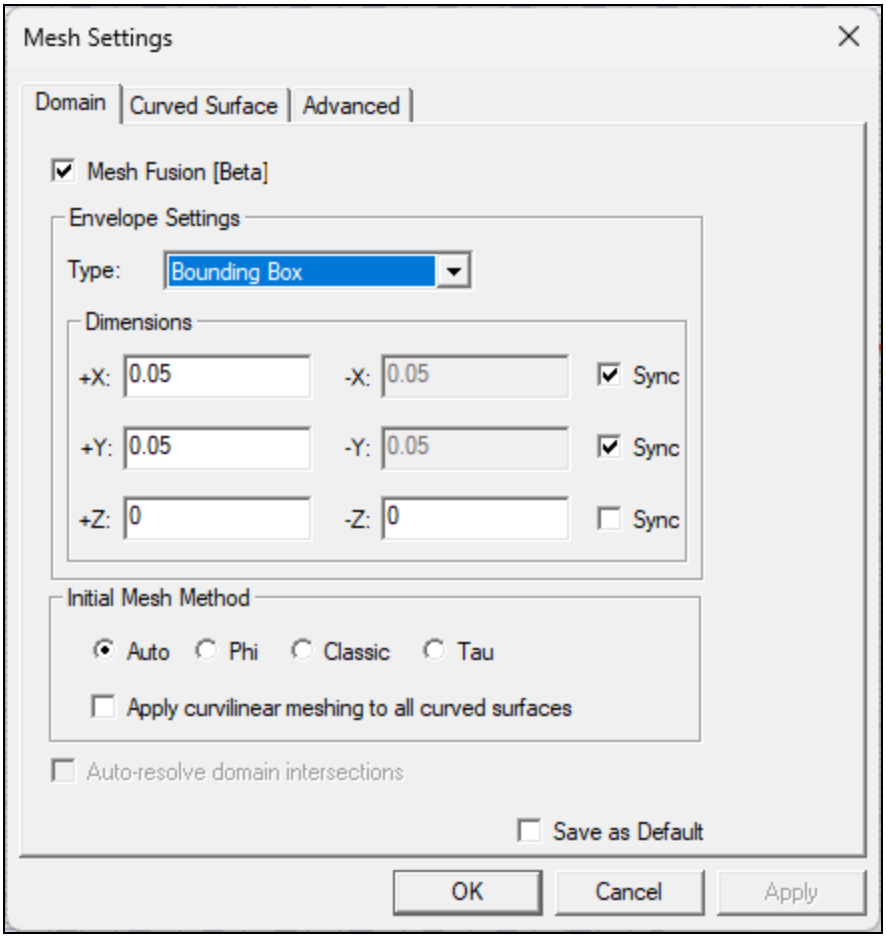
Care should be taken when defining the domain envelopes to ensure:

- There is no overlap between native and component geometry.
- There is no overlap between component envelopes.
- Envelope definitions contain only planar faces.
- Envelope boundaries are not coincident with ports.
- Envelope definitions provide for optimal initial meshing.
    - Insulate geometry not compatible with the Phi mesher from geometry that is compatible.
    - Mesh geometry of different scales in their own domain.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Method Override

The initial mesh method for a given component instance can be selected in the **Mesh Settings** window. This allows for the optimal mesh technology to be applied on a component-by-component basis. Refer to "Mesh Setup for Mesh Fusion/Assembly" on page 17-26.

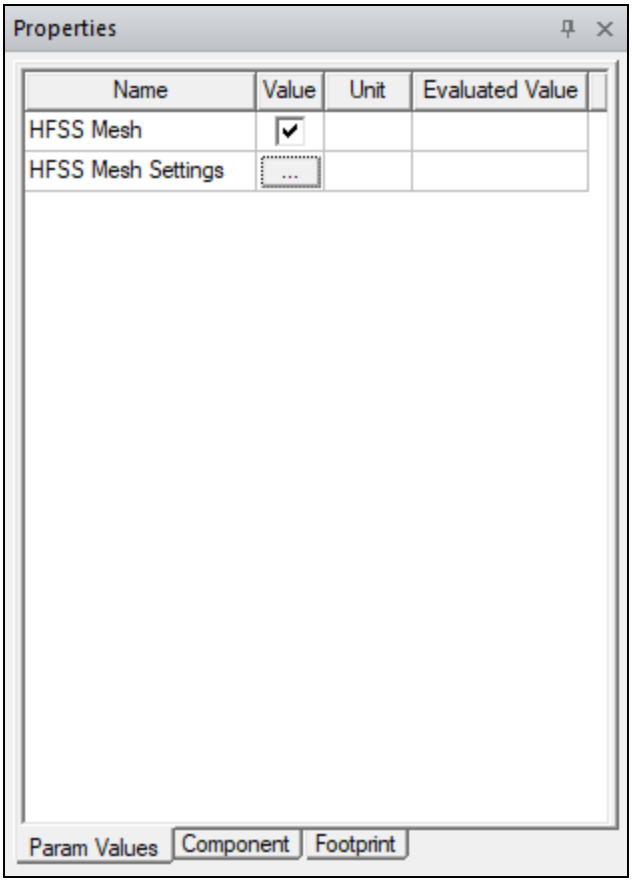


Access the Mesh Settings Window

To access the **Mesh Settings** window, follow these steps.

1. From the **Layout Editor**, select a 3D object in the active design to make its parameters appear 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.



2. If applicable, click on the **Param Values** tab in the **Properties** window.

> **Note:**
>
> Mesh Fusion is supported **only** for layout, 3D, and 2.0 components. If a component does not meet this requirement for Mesh Fusion, the option is not available on the **Param Values** tab.

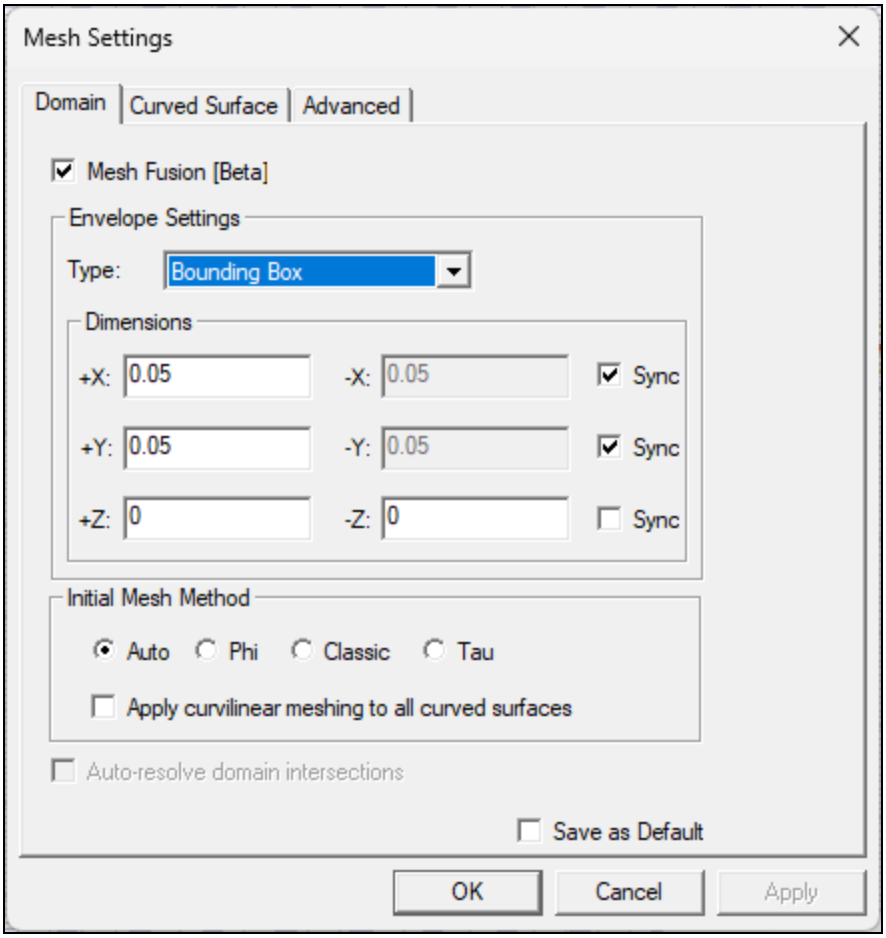
3. Click the **HFSS Mesher Settings [...]** button to open the **Mesh Settings** window with *Domain*, *Curved Surface*, and *Advanced* tabs.

Some of the features of Mesh Fusion include:

- Systems with large variation in length scales can be successfully meshed (e.g., in the case of an IC mounted on a package, mesh the components separately and with optimal pre-processing for their respective scale).
- Different meshers can be selected for each component via the **Initial Mesh Method** area (i.e., choose between **Auto**, **Phi** or **Phi Plus** for an ECAD component, and **Classic** or **Tau** for an MCAD 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:**
>
> The Tau mesher is a beta feature. Beta features must be activated prior to use. Refer to Activating the Tau Mesher Beta Feature on the Initial Meshing Method page for instructions.



Mesh Fusion/Assembly for Hierarchical Design

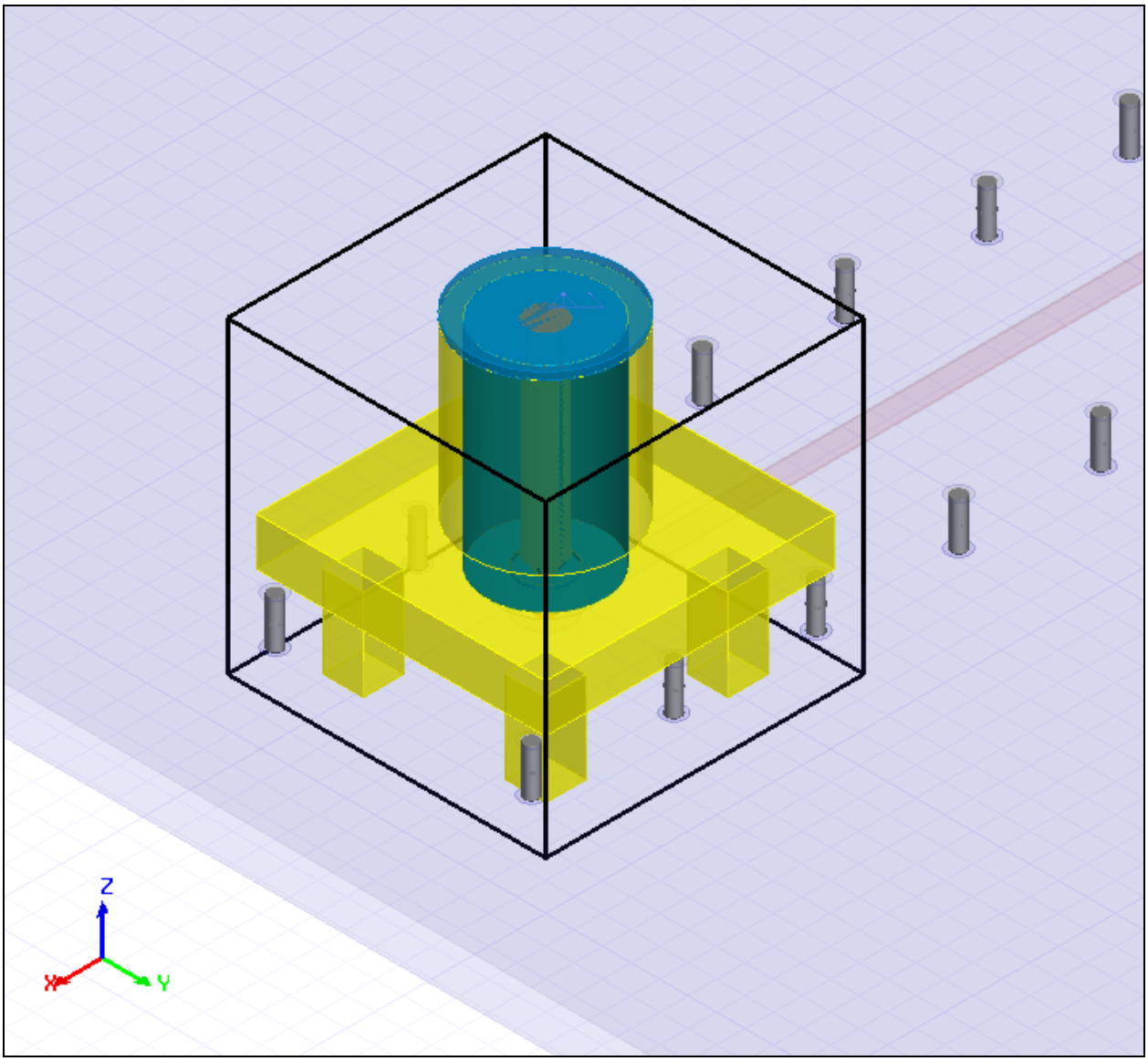
In HFSS 3D Layout, a hierarchical design is defined by the top-level layout and a collection of component instances. Component instances can be of different types, most commonly other components, 3D Components (MCAD), and Component 2.0 representations. For mesh assembly/fusion (Mesh Fusion), this hierarchy is used to define independent mesh domains, with then *n*top-level layout referred to as the “native” region. In allowing separate regions to be meshed independently, Mesh Fusion enables robust and rapid initial meshing.

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 use Mesh Fusion, the native region must encapsulate all component instances. For a component instance to be modeled as a separate mesh domain, check the **Mesh Fusion** box in the "Access the Mesh Settings Window" on page 7-393 window.

By default, components selected for Mesh Fusion are assigned a bounding-box envelope for its mesh region. If a 3D component has explicitly defined “mesh regions”, those are also available for envelope assignment.



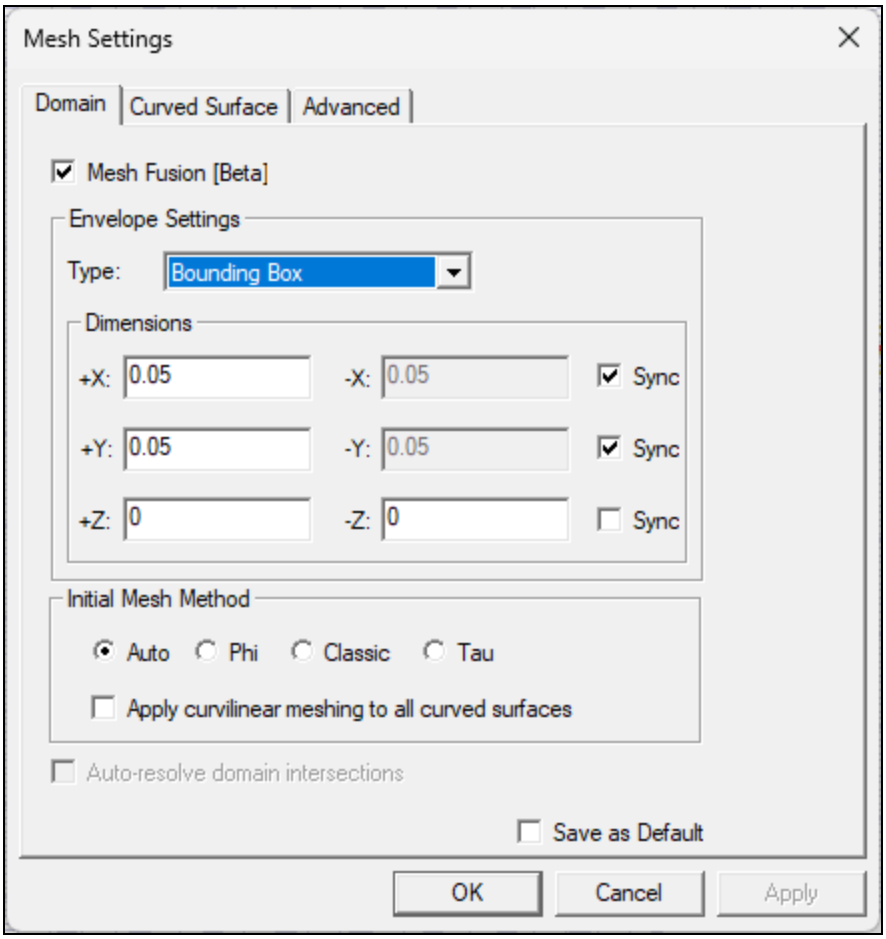
Care should be taken when defining the domain envelopes to ensure:

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 no overlap between native and component geometry.
- There is no overlap between component envelopes.
- Envelope definitions contain only planar faces.
- Envelope boundaries are not coincident with ports.
- Envelope definitions provide for optimal initial meshing.
    - Insulate geometry not compatible with the Phi mesher from geometry that is compatible.
    - Mesh geometry of different scales in their own domain.

Mesh Method Override

The initial mesh method for a given component instance can be selected in the **Mesh Settings** window. This allows for the optimal mesh technology to be applied on a component-by-component basis. Refer to "Mesh Setup for Mesh Fusion/Assembly" on page 17-26.

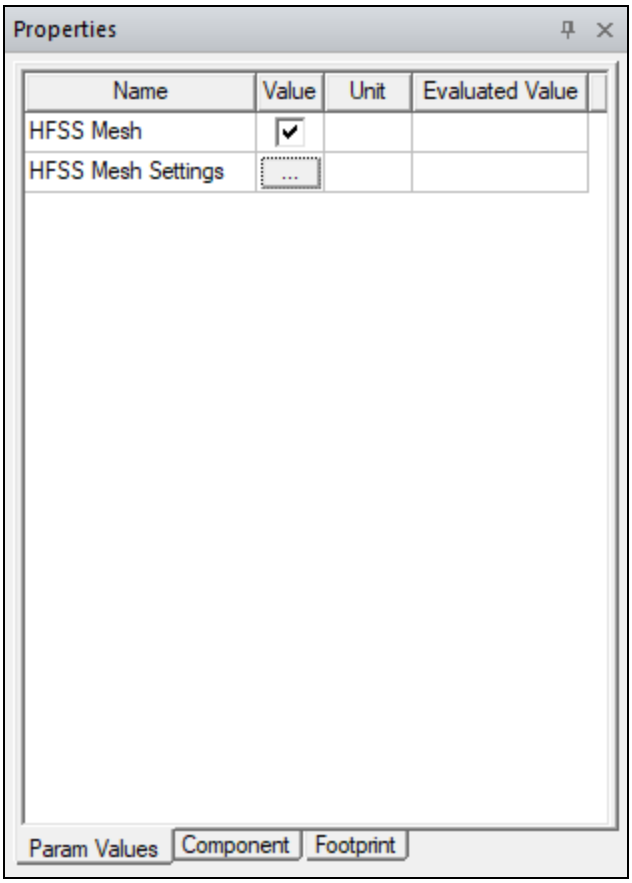


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Access the Mesh Settings Window

To access the **Mesh Settings** window, follow these steps.

1. From the **Layout Editor**, select a 3D object in the active design to make its parameters appear in the **Properties** window.



2. If applicable, click on the **Param Values** tab in the **Properties** window.

> **Note:**
>
> Mesh Fusion is supported **only** for layout, 3D, and 2.0 components. If a component does not meet this requirement for Mesh Fusion, the option is not available on the **Param Values** tab.

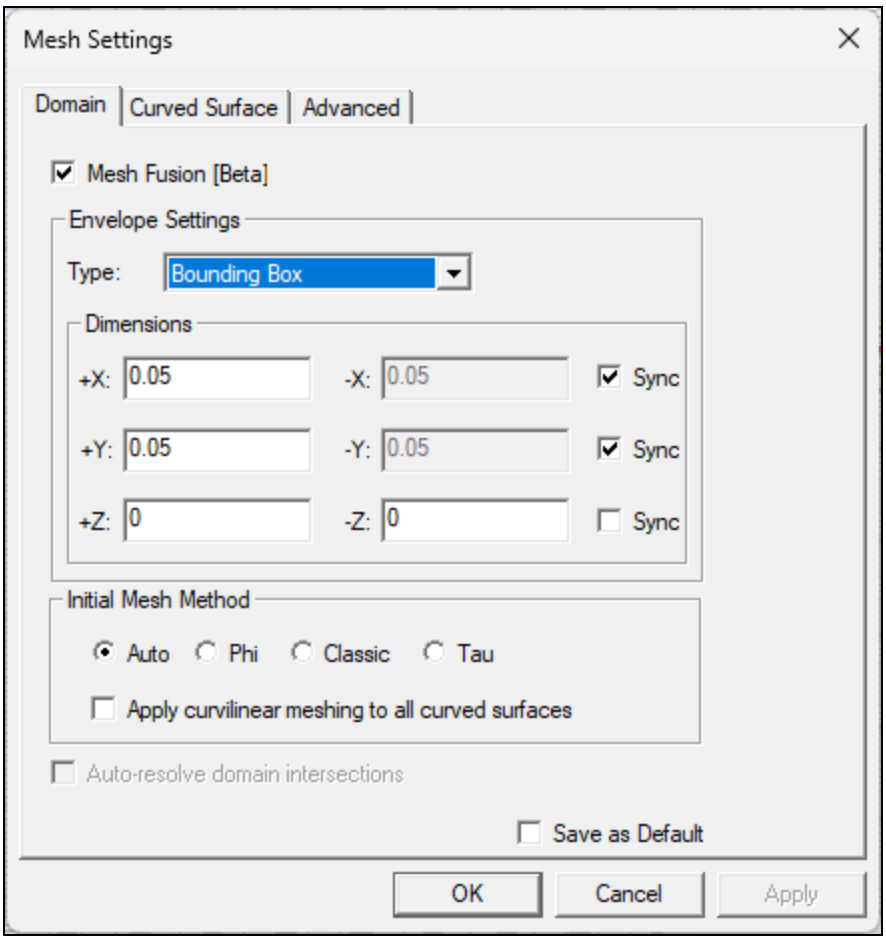
3. Click the **HFSS Mesher Settings [...]** button to open the **Mesh Settings** window with *Domain*, *Curved Surface*, and *Advanced* tabs.

Some of the features of Mesh Fusion include:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Systems with large variation in length scales can be successfully meshed (e.g., in the case of an IC mounted on a package, mesh the components separately and with optimal pre-processing for their respective scale).
- Different meshers can be selected for each component via the **Initial Mesh Method** area (i.e., choose between **Auto**, **Phi** or **Phi Plus** for an ECAD component, and **Classic** or **Tau** for an MCAD component).

> **Note:**
>
> The Tau mesher is a beta feature. Beta features must be activated prior to use. Refer to Activating the Tau Mesher Beta Feature on the Initial Meshing Method page for instructions.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Check

Running Geometry Check before running an analysis on a model can find and automatically fix common problems that may prevent a simulation from completing successfully.

To perform a geometry check on the active project:

1. Click **HFSS 3D Layout** > **Geometry Check** to open the **Launch Geometry Check** window. Select any checks that you want to validate:
   - Select **Self-Intersecting Polygons** to check for duplicating planes, intersecting edges, and check whether voids intersect the plane boundaries.
   - Select **Disjoint Nets** to check if there are nets that are not completely connected.
   - Select **DC-Shorted Errors** to check if any nets overlap.
   - Select **Identical/Overlapping Vias** to check if any vias are identical or overlap on the same nets.
   - Select **Misalignments** to check for all misalignments in the following cases:
   - Flag and auto-correct areas where the centerline of a trace is not contained within a plane.
   - Check if a trace overlaps with a plane, but the trace centerline does not intersect with a plane.
   - Check if a trace overlaps with another trace, but the centerline of a trace does not intersect with the centerline of another trace.
   - Check if the boundary of a pad/via intersect with the boundary of a trace, but the center of a via/pad is not located on the centerline of a trace (in other words, the pad/via is not snapped to the centerline of a trace).
   - Check if the boundary of a plane intersect with the boundary of a pad/via, but the center of the pad/via is not contained in the boundary of the plane.
   - Check if a trace overlaps with a plane, but the trace centerline does not intersect with a plane.
   - Check if a trace overlaps with another trace, but the centerline of a trace does not intersect with the centerline of another trace.
   - Check if the boundary of a pad/via intersect with the boundary of a trace, but the center of a via/pad is not located on the centerline of a trace (in other words, the pad/via is not snapped to the centerline of a trace).
   - Check if the boundary of a plane intersect with the boundary of a pad/via, but the center of the pad/via is not contained in the boundary of the plane.
   - Check if a trace overlaps with a plane, but the trace centerline does not intersect with a plane.
   - Check if a trace overlaps with another trace, but the centerline of a trace does not intersect with the centerline of another trace.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Check if the boundary of a pad/via intersect with the boundary of a trace, but the center of a via/pad is not located on the centerline of a trace (in other words, the pad/via is not snapped to the centerline of a trace).
- Check if the boundary of a plane intersect with the boundary of a pad/via, but the center of the pad/via is not contained in the boundary of the plane.

The following checks are done in all cases, regardless of the selections you make:

Zero Via Plating — Flags padstack definitions with no (0%) plating.

Pin Shared By Multiple Pin Groups — Flags pins that belong to more than one pin group.

2. Click **OK** in the **Launch Geometry Check** window.

   The progress bar displays the progress. The **Geometry Check Results** window opens and lists errors and warnings. Items that can be fixed automatically contain **Auto Fix** check boxes; select a box to have a problem automatically corrected.

3. Click **OK** in the **Geometry Check Results** window.

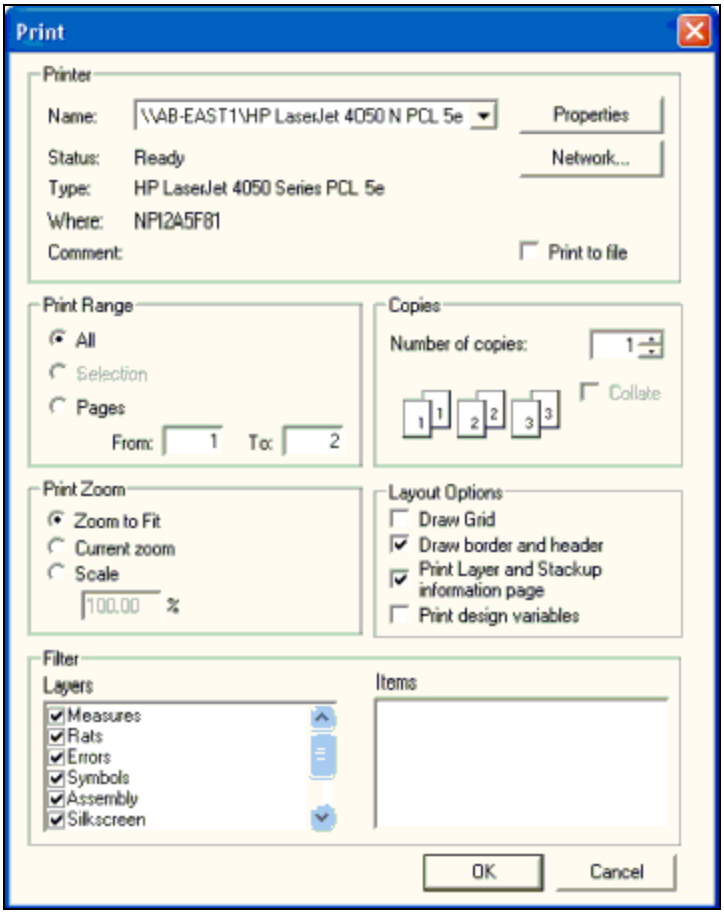
   The result details can be viewed in the **Message Manager**. Most geometry check results are displayed with a magnifying glass that can be clicked to zoom into the relevant area of the design. It may help to toggle layer and net visibilities when zooming into individual results.

Printing a Layout

To print the layout that is in the 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.

1. Select **File** > **Print** to open the Print window:



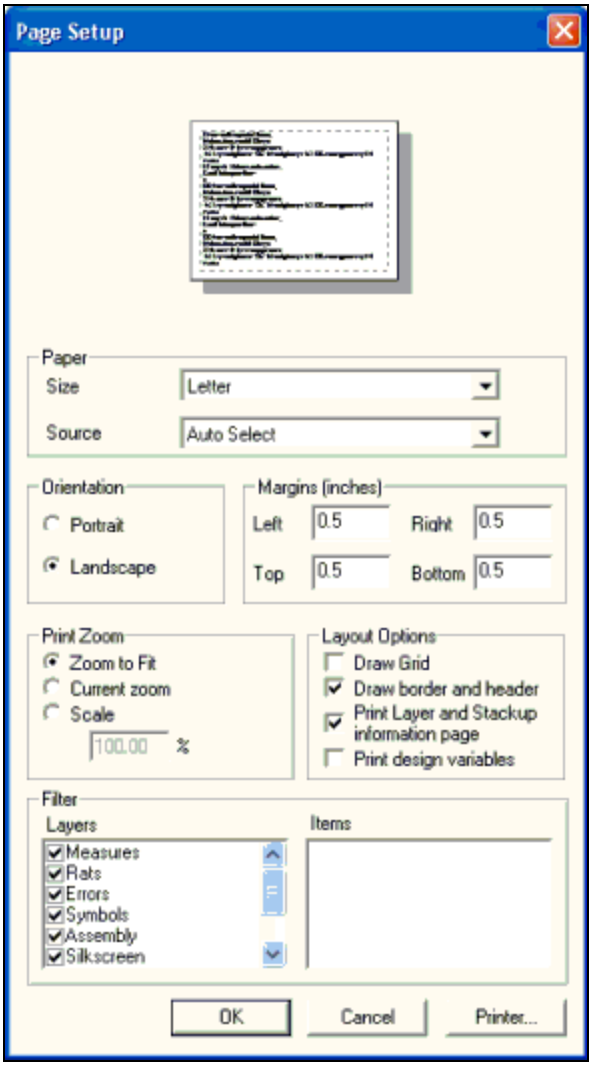
2. Use the **Print Range**, **Copies**, and **Print Zoom** controls to specify print preferences. From the **Layout Options** area, specify whether to draw the grid, border/header, the layer and stackup information, and design variables. From the **Filter** area, toggle the printing of any layer and selected items.
3. Click **OK** to print the layout and close the **Print** window.

Setting up the Page Layout

To set up the page layout before printing:

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 **Page Setup** on the **File** menu to open the **Page Setup** window.



2. Use the **Paper**, **Orientation**, **Margins**, and **Print Zoom** controls to specify print preferences. From the **Layout Options** area, specify whether to draw the grid, border/header, layer and stackup information, and design variables. From the **Filter** area, toggle the printing of any layer and selected items.
3. Click **OK** to close the **Page Setup** window with your settings.
4. Select **Print Preview** on the **File** menu to see a preview of the print.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8 - Assigning Materials

You can add, remove, and edit materials in two main ways:

- Using the **Tools** > **Edit Libraries** > **Materials** menu command.
- Under **Definitions** in the Project Manager, right-click **Materials** and select **Edit Library**.

Regardless of which of the preceding two methods you use to edit a library, any new material you create exists only in the current project. Similarly, if you edit an existing material, the edited version becomes a *Project* material and exists only within the current project. To make a new or modified material available for other projects, you must export it to a user library and choose that library (or select **Show all libraries**) within the **Select Definition** dialog box.

> **Note:**
>
> In the History Tree, by default, objects are grouped by material. You can easily change the default grouping. Using the menu bar, click **Modeler** > **History Tree Layout** and clear **Group Objects by Material** along with any other listed option you may wish to disable. You can also perform this action from the **Tree View** drop-down menu on the **View** ribbon tab or from the shortcut menu that appears when you right-click most of the items in the History Tree.

If you check **Show Material Colors** on the Material Filters tab of the **Select Definition** or **Edit Library** dialog box, the **Materials** tab will include a **Color** column showing a color swatch for each listed material.

You can edit the color and transparency values for materials in the View/Edit Material window.

Solving Inside or on the Surface

When you assign a material to an object, you can specify whether to generate a field solution inside the object or on the surface of the object. If you elect to generate a solution inside the object, a mesh is created inside the object and generate a solution from the mesh. If you elect to generate a solution on the surface of the object, only a surface mesh is created for the object.

If you want a solution to be generated inside an object, select **Solve Inside** in the **Properties** window. Conversely, if you want a solution to only be generated on the surface of an object, clear the **Solve Inside** option in the **Properties** window.

Assigning DC Thickness

The **Assign DC Thickness** option allows you to more accurately compute the DC resistance of a thin conducting object for which Solve Inside is not selected. Skin impedance of the object will

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

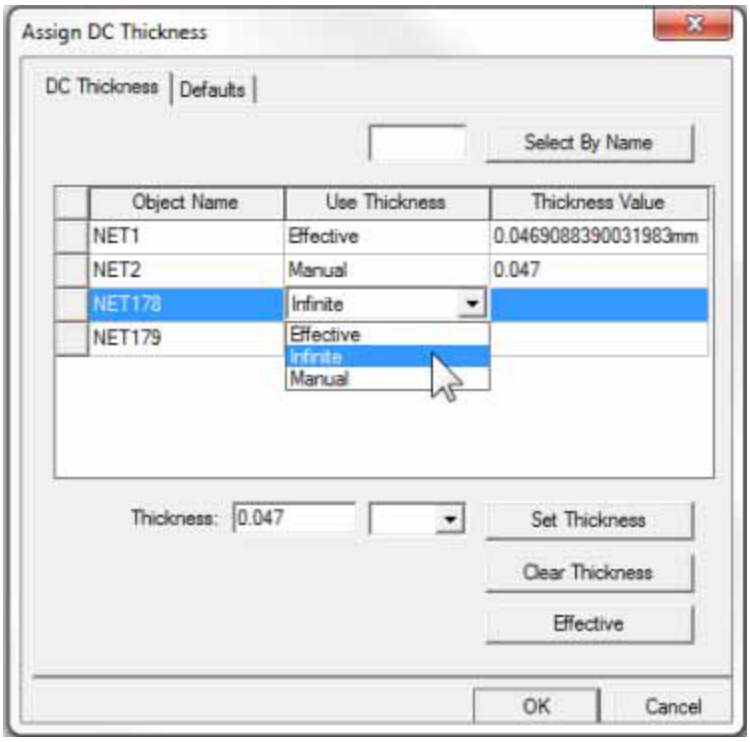
be calculated using the defined finite thickness, which can be Automatic (the default), Manually assigned per object, or Infinite per object. This option also exists for finite conductivity boundaries.

The **Assign DC Thickness** option on the **Boundaries** menu is enabled if at least one object contains a good conducting isotropic material (such as copper), and the **Solve Inside property** is not selected. If the object meets these conditions, you can assign a DC thickness, either by enabling the automatic default, or by specifying a value for a selected object.

To see the **Assign DC Thickness** dialog box:

1. Select **HFSS 3D Layout** > **Boundaries** > **Assign DC Thickness**.

   This displays the **Thickness of Objects for DC Resistance** dialog box with the DC Thickness tab selected. Objects to which the thickness can be applied are listed in the **Object Name** column.



2. Select the objects to assign a value. You can select objects either by:

- Clicking on the **Object Name** to highlight it.
- Use the **Select By Name** field to type the object name, and click **Search**.

  The first object to match the name is highlighted.

Selecting an object highlights the **Thickness** field and the **Set Thickness** button.

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 Use Thickness column, you can specify that the value the object uses is Automatic, Infinite, or Manual.

You can disable automatic assignment on the **Defaults** tab of the **Assign DC Thickness** dialog box. The Automatic value is calculated as Thickness ~ 2* Volume / Surface Area

It should be noted that this is a calculation for an "effective" DC thickness to be used by the correction calculation. For a "thin" object this will work well. For example, a rectangular microstrip trace described by a box with dimensions 100 by 10 by 1 the volume is 1000 and the surface area is 2220 resulting in an apparent thickness of .9009, close to the geometric thickness of "1". For arbitrary shapes of "thicker" objects, this calculation will not work as well. For example, a cube with sides of 1 will have volume of 1 and surface area of 6, and a resulting apparent thickness of .3333. Another example is the case of a cylindrical wire (e.g., bond wire).

In this case, the automatic effective DC thickness will be about R_o, which gives us the best approximation of the DC resistance of a cylindrical wire.

The intention is that the auto-thickness will provide an accurate representation the majority of the time and is superior to not using any DC thickness setting. When the automatic value is not appropriate, you can override it using the manual technique.

DC thickness impedance is an approximation. It is accurate just for TE/TM waves when the widths are infinite which clearly never occurs in a "real" design. The fact that the object is finite causes an increase of the effective impedance due to current crowding/edge effects. Thus entering the exact geometrical thickness would actually underestimates the impedance. By returning a DC thickness smaller than the geometric thickness the automatic DC thickness compensates for this underestimation resulting in a slightly higher impedance as desired.

4. To manually apply a value, enter a **Thickness** value, select the units and click the **Set Thickness** button.

This applies the value to the selected object and changes the **Use Thickness** selection for that object to **Manual**.

Note:

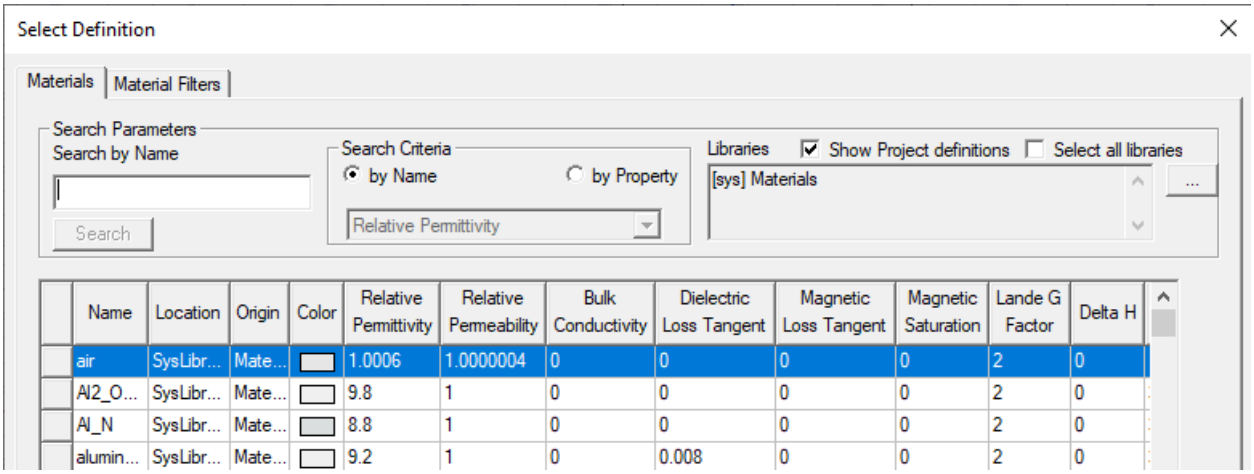
If you enter a “0” for the thickness, HFSS gives a warning that this will cause infinite impedance that causes isolation.

5. To change the value, select the **Clear Thickness** button and then enter a different value. You can also manually select or deselect the box and manually enter or delete a thickness value in the table.
6. When you have assigned the values you need, click **OK** 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.

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.

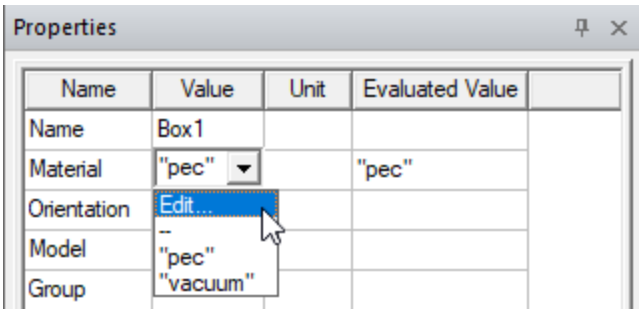


To search for a material **by Name**:

1. Access the **Select Definition** dialog box using one of the following methods:

   With one or more objects selected (that is, to assign a material to selected objects):

   - Click **Modeler** > **Assign Material**
   - Click the **Material** *value* in the docked **Properties** window, and select **Edit...** from the drop-down menu.

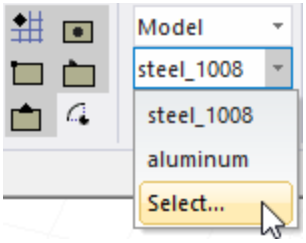


   - On the **Draw** ribbon tab, click **Assign material**.

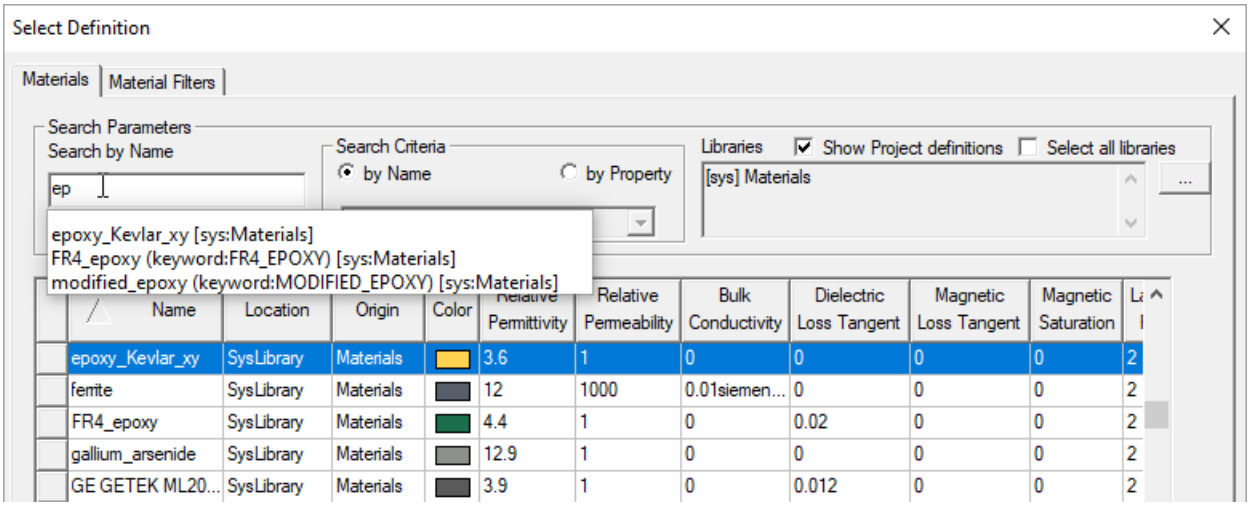
   With nothing selected (that is, to set the default material):

   - On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Search Criteria** section, ensure that **by Name** is selected, and in Libraries, specify the Libraries that you want to search. Only loaded libraries participate in text and keyword matching.
3. In the **Search by Name** text box, type a portion of the desired material name. The search text is case-insensitive.



A drop-down 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:

- 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data for Simulation library now includes keywords that are used in the auto-complete matching. For example, entering "glass" will generate an auto-complete list of all materials with a name containing "glass" or with a keyword that contains "glass", all case-insensitive, for materials showing in the grid. The drop down of potential matches under the search field applies to all libraries. What shows in the grid is defined by which libraries are selected, what material filters are selected, project/design-specific validation, and whether project materials are selected for display.

Searching by Material Property

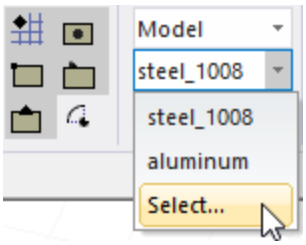
1. Access the **Select Definition** dialog box using one of the following methods:

   With one or more objects selected (that is, to assign a material to selected objects):

   - Click **Modeler** > **Assign Material**.
   - Click the **Material** *Value* in the docked *Properties* window, and select **Edit...** from the drop-down menu.
   - On the **Draw** ribbon tab, click **Assign material**.

   With nothing selected (that is, to set the default material):

   - On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:



2. In the **Search Criteria** section, select **by Property**.
3. Select a material property from the 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**.

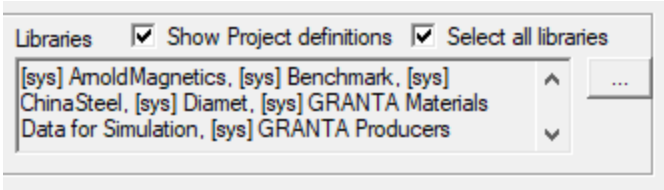
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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**. 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**.

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/Edit Material** window appears.

By default, only properties commonly used by the selected product are displayed. To view the complete table of properties, see Filtering Materials.

2. Type a name for the material in the **Material Name** text box, or accept the default.
3. Use the radio buttons in the **View/Edit Material for** section to specify whether the new materials apply to Active Design, Active Project, or All Properties. When **All Properties** is selected, **Physics** classification options are enabled to show or hide properties based on simulation type (Electromagnetic, Thermal, or Structural).

> **Note:**
>
> If a material is edited in a design type for which the **Physics** type has not been set (e.g., an design but physics type was not set), the **Physics** type will be automatically set in the material.

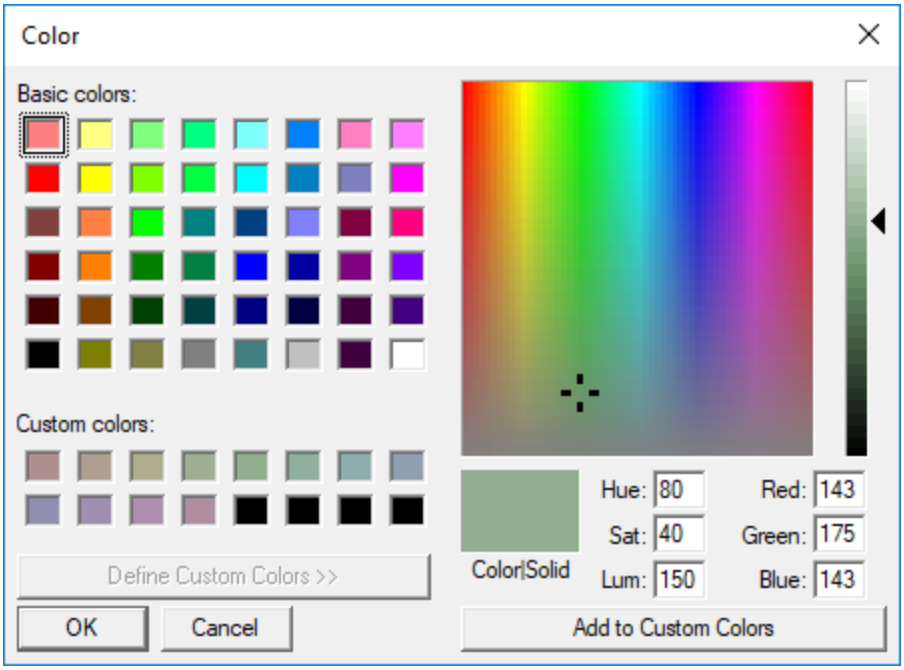
You can also enable the View/ Edit Modifier check box for Thermal Modifier. Checking this box causes the Thermal Column to display at the right side of the Properties of the Material table. Selecting **Edit** rather than None causes display of the Edit Thermal Modifier dialog.

4. Enter a **Material Name**, or accept the default.
5. For each available material property, select a **Type** from the drop-down menu. Only applicable types appear for each material property. Some properties only use the Simple type, while others offer four or more types.
6. nter a **Value** for the following material properties:

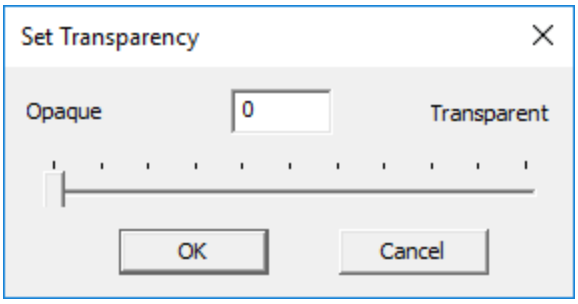
> **Note:**
>
> You may enter a variable name or mathematical expression in the **Value** box.

7. If one or more of the material properties are dependent on frequency, click **Set Frequency Dependency**, and then follow the directions for defining frequency dependent materials.
8. To modify the units for a material property, double-click the **Units** box and select a new unit system.
9. For Material Appearance, you can check the box to enable the fields for you to specify a color and transparency. Clicking the color bar opens a color selection window:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

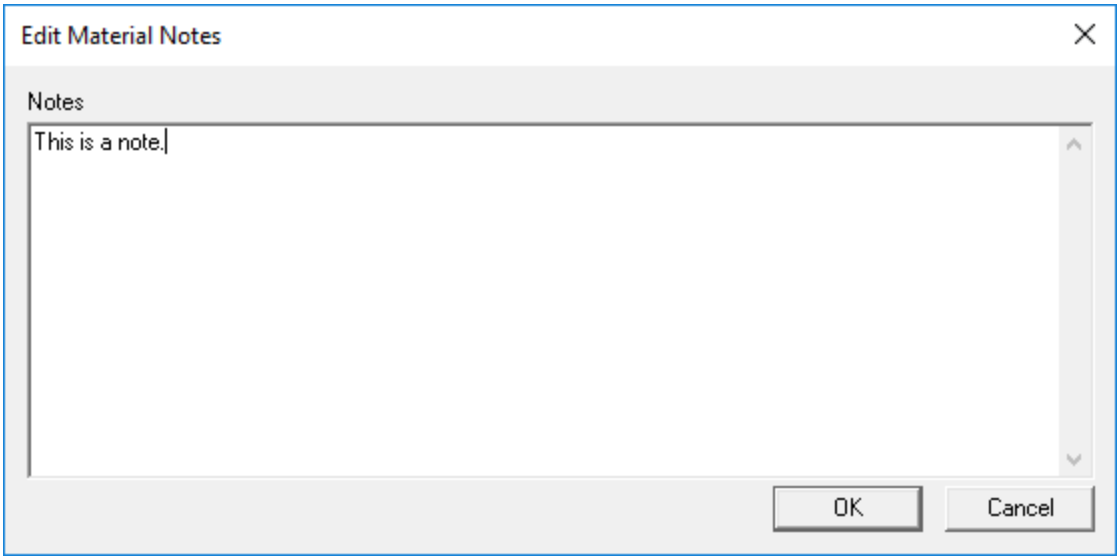


10. Clicking the Transparency box opens a Transparency dialog box with a text field and slider bar for selection.



11. Click **OK**.
12. To modify the units for a material property, double-click the **Units** box and select a new unit system.
13. Click **OK**.
14. If you want to add descriptive notes for the new material, click the ellipsis button [...] next to the **Notes** field. This opens a dialog box in which you can enter text.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



15. Click **OK** to add the Notes.
16. Click **OK** on the **View/Edit Materials** 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:

Simple	The material is homogeneous and linear.
Anisotropic	The material's characteristics vary with direction. This models only the diagonal.
Tensor	A full 3x3 matrix general tensor, which may or may not be symmetrical.
Nonlinear	A BH Curve.

Select a material property type for each property from the **Type** drop-down menu. Some properties only use the Simple type

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic . Each diagonal represents a tensor of your model along an axis.

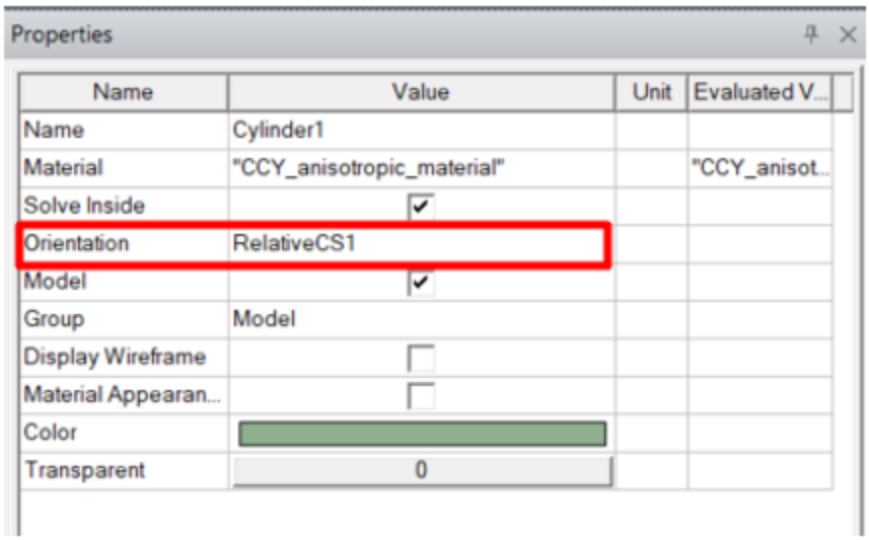
These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 :

1. From the **View/Edit Material** window, use the **Type** drop-down menu to select **Anisotropic**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 . along each axis of the material's tensor.
4. Click **OK** to save the values and return to the **Select Definition** window.

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 and define its value as **4**. To assign this property value to a material, type in the 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.

Defining Frequency-Dependent Material Properties

HFSS provides several frequency-dependent material models. The Piecewise Linear and Frequency Dependent Data Points models apply to both the electric and magnetic properties of the material. However, they do not guarantee that the material satisfies causality conditions, and so they should only be used for frequency-domain applications.

The Debye and Djordjevic-Sarkar models apply only to the electrical properties of dielectric materials. These models satisfy the Kramers-Kronig conditions for causality, and so are preferred for applications (such as TDR or Full-Wave Spice) where time-domain results are needed. The Design Settings also include an automatic Djordjevic-Sarkar model to ensure causal solutions when solving frequency sweeps for simple constant material properties.

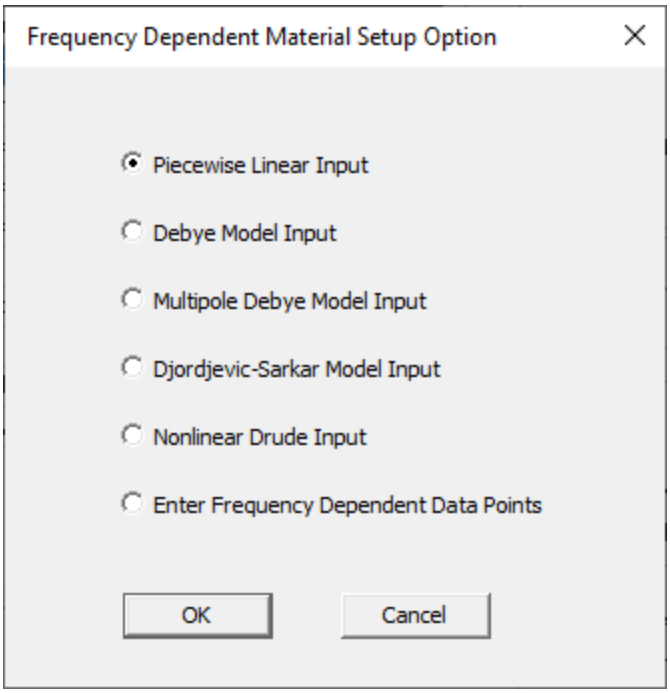
In HFSS, you can assign conductivity either directly as bulk conductivity, or as a loss tangent. This provides flexibility, but you should only provide the loss once. The solver uses the loss values just as they are entered.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Frequency-Dependent Properties

1. From the **Select Definition** window, select a material and click **View/Edit Material**.
2. Click **Set Frequency Dependency**.

   The **Frequency Dependent Material Setup Option** dialog box appears.



The options are:

- **Piecewise Linear Input** – defines the material property values as a restricted form of piecewise linear model with exactly 3 segments (flat, linear, flat). You specify the property's values at an upper and lower corner frequency. Between these corner frequencies, HFSS linearly interpolates the material properties; above and below the corner frequencies, HFSS extrapolates the property values. This dataset can be modified with additional points if desired.
- **Debye Model Input** – a single-pole model for the frequency response of a lossy dielectric material. In some materials, up to about a 10-GHz limit, ion and dipole polarization dominate and a single pole Debye model is adequate. HFSS allows you to specify an upper and lower measurement frequency, and the loss tangent and relative permittivity values at these frequencies. You may optionally enter the permittivity at optical frequency, the DC conductivity, and a constant relative permeability.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Multipole Debye Model Input** – allows you to provide the data of relative permittivity and loss tangent versus frequency. Based on this data, the software dynamically generates frequency-dependent expressions for relative permittivity and loss tangent through the Multipole Debye Model. The input dialog plots these expressions together with your input data through the linear interpolations. The generated expressions provide the new value for the material properties of relative permittivity and loss tangent. Both the expressions and data triples can be saved and reloaded.
- **Djordjevic-Sarkar Model Input** – for low-loss dielectric materials (particularly FR-4) commonly used in printed circuit boards and packages. In effect, it uses an infinite distribution of poles to model the frequency response, and in particular the nearly constant loss tangent, of these materials. This allows you to enter the relative permittivity and loss tangent at a single measurement frequency. You may optionally enter the relative permittivity and conductivity at DC.
- **Nonlinear Drude Input** – Drude materials are now supported by the transient solvers, producing plasma density data within the material domains. This means that plasma density plots can be produced.
- **Enter Frequency Dependent Data Points** – allows you to enter, import or edit frequency-dependent data sets for each material property. Any number of data points may be entered. This is an arbitrary piecewise linear model.

3. Select the input type and click **OK**.

   An input type-specific dialog box appears, based on your selection.

   Piecewise Linear Frequency Dependent Material Input, Debye Model Input, and Djordjevic-Sarkar Model Input remember the values previously used and also include plots to show the property curves in real time as changes to the input are made. Input values for each are saved as material attached data for the material being edited. These data items are saved with materials when they are written into a project file or exported to a material library. Note that when a frequency-dependent setup method is used and the values are prepopulated with saved data, the dialog box title will have "(Update)" appended.

After you have entered the data for your selection, you return to the **View/Edit Material** window. New default function names appear in the material property text boxes. HFSS automatically creates a dataset for each material property. Based on a varying property's dataset, HFSS can interpolate the property's values at the desired frequencies during solution generation.

To modify the dataset with additional points, see *Editing Datasets*.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Neither the piecewise nor the loss model asks for frequency-dependent conductivity because the constant sigma represents DC loss and the frequency-dependent loss tangent represents polarization losses.

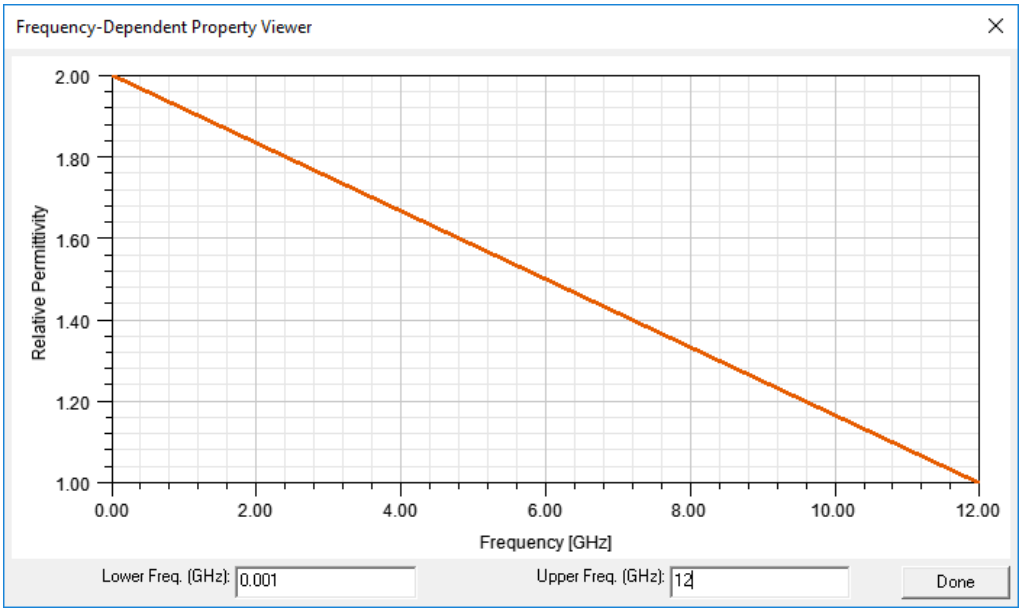
Frequency Dependence Visualization

When you view or edit material properties, it is important to have a sense of how properties may vary with frequency. Frequency-dependent properties come in a variety of forms, ultimately resulting in some value expression or dataset. Plots of properties as a function of frequency are available through the **View/Edit Materials** window.

To view the plot:

1. From the **Select Definition** window, select a material and click **View/Edit Material**.
2. Right-click a property (for example, relative permittivity or dielectric loss tangent), and select **View Property vs. Frequency**.

   The **Frequency-Dependent Property Viewer** appears, displaying the plot.



The window contains boxes where you can edit the frequency range for the plot. If the property was set by one of the nonlinear frequency setup methods, the frequency range will be derived from the data for that method, and edits to the lower/upper frequencies are not saved. Otherwise, the frequency range lower/upper frequency limit defaults are stored in the registry, and are updated if you modify the values. If values are not yet stored in the registry, 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.

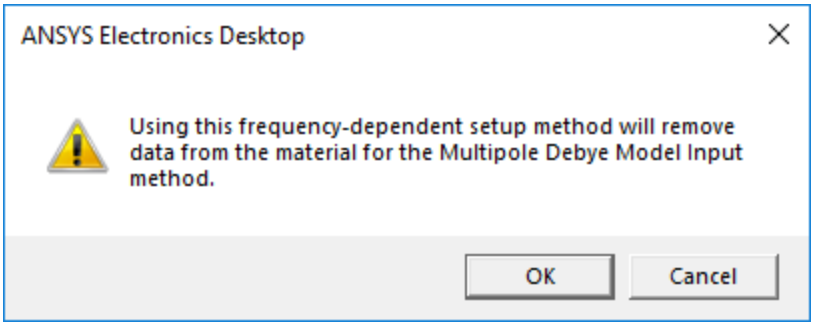
defaults to 1MHz-10GHz. Note that **the View Property vs. Frequency** menu option is not displayed for choice properties because frequency-dependence doesn't apply to those.

Saved Input Data Invalidation For Frequency-Dependent Setup

You will be prompted with a warning in two circumstances:

- Data associated with one of the frequency-dependent setup methods is attached to the material definition, and a property which would be set by this method is modified.
- Data associated with one of the frequency-dependent setup methods is attached to the material definition, and a different setup method is subsequently used.

You can choose **OK** to continue with the edit and remove the invalidated setup data, or choose **Cancel** to discard any changes.

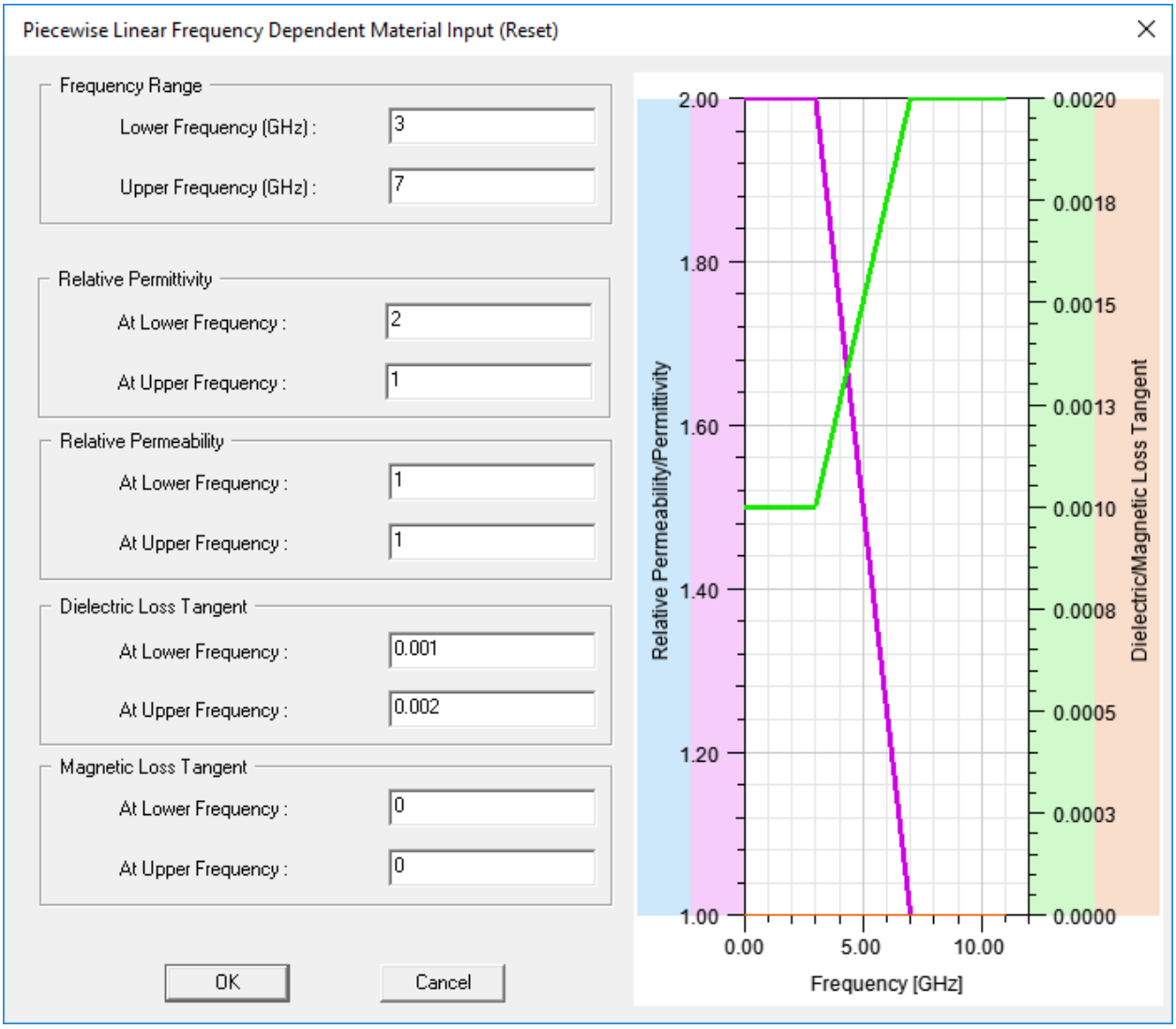


Assigning Frequency Dependent Material: Piecewise Linear Input

When you select **Piecewise Linear Input** as the model for the frequency dependent material property, the **Piecewise Linear Frequency Dependent Material Input** window appears.

From this window, you can enter material property values and view a property vs. frequency plot.

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 to enter lower and upper frequency values for:

- **Frequency Range** – assumes that the material's property values remain constant between these frequencies.
- **Relative Permittivity** – If the permittivity of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.
- **Relative Permeability** – If the permeability of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.
- **Dielectric Loss Tangent** – If the dielectric loss tangent of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.
- **Magnetic Loss Tangent** – If the magnetic loss tangent of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

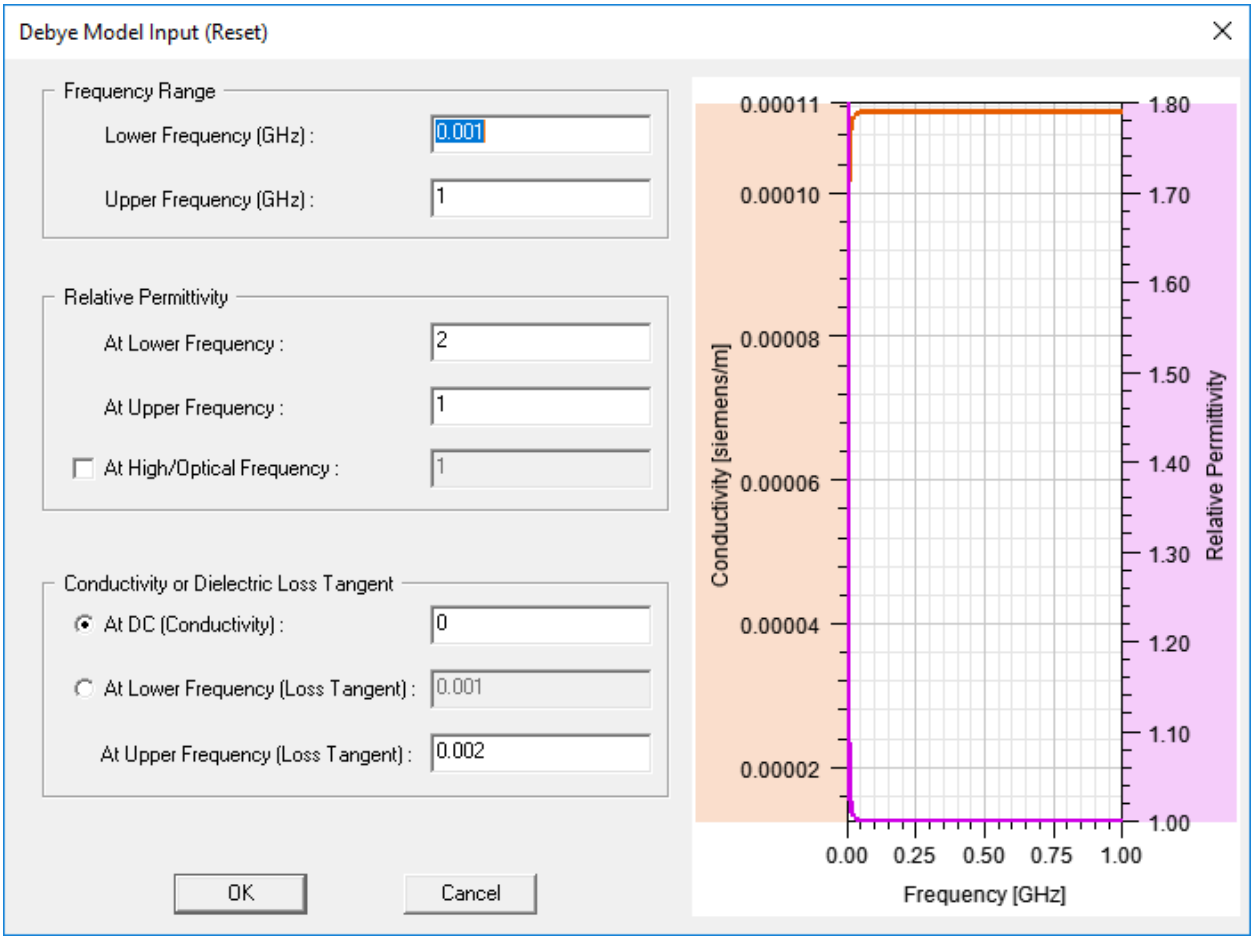
> **Note:**
>
> Neither the piecewise or the loss models ask for frequency dependent conductivity because there the constant sigma represents the DC loss and the frequency dependent loss tangent represents the polarization losses.

To modify the dataset with additional points, see Editing Datasets.

Assigning Frequency Dependent Material: Debye Model Input

When you select **Debye Model Input** as the model for the frequency dependent material property, the **Debye Model Input** window appears.

From this window, you can enter material property values and view a property vs. frequency plot.



Use the fields to enter lower and upper frequency values for:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Frequency Range** – assumes that the material's property values remain constant between these frequencies.
- **Relative Permittivity** – If the permittivity of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency. If you need to specify a value for At **High/Optical Frequency**, select the check box and enter a value.
- **Conductivity or Dielectric Loss Tangent** – Select either **At DC** to specify conductivity, or **At Lower Frequency** to specify Loss Tangent.

After you have entered the data for your selection, click **OK** to return to the **View/Edit Material** window. New default function names appear in the material property text boxes. A dataset is automatically created for each material property. Based on a varying property's dataset, can interpolate the property's values at the desired frequencies during solution generation.

To modify the dataset with additional points, see Editing Datasets.

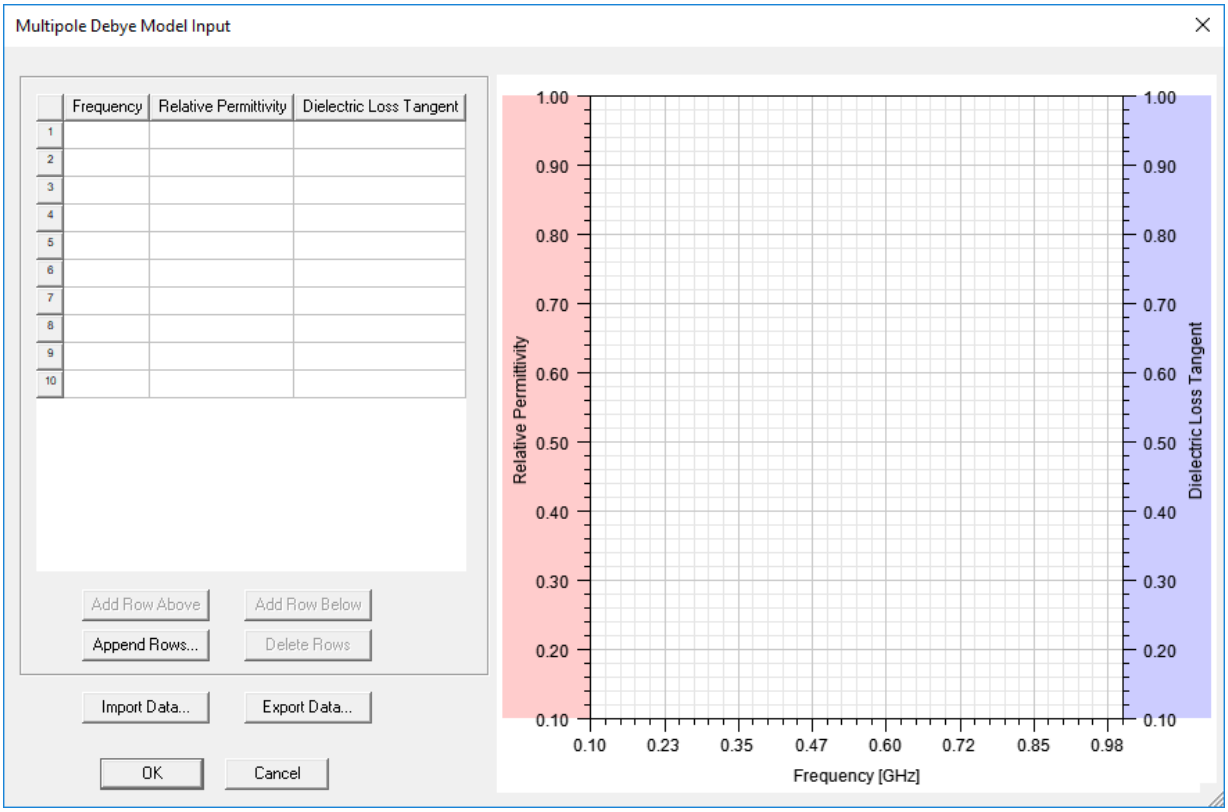
> **Note:**
>
> Neither the piecewise or the loss models ask for frequency dependent conductivity because there the constant sigma represents the DC loss and the frequency dependent loss tangent represents the polarization losses.

Assigning Frequency Dependent Material: Multipole Debye Model Input

When you select **Multipole Debye Model Input** as the model for the frequency dependent material property, the **Multipole Debye Model Input** window appears.

From this window, you can enter material property values and view a property vs. frequency 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 window contains a table that allows you to specify **Frequencies**, and the material's **Relative Permittivity** and **Dielectric Loss Tangent** at each frequency.

Note:

- The minimum value for Frequency is 0. There is no upper limit.
- The minimum value for Relative Permittivity is 1. There is no upper limit.
- The minimum value for Dielectric Loss Tangent is 0. There is no upper limit.
- To get good results, provide at least 5 frequency points.

Enter data values manually, or click **Import Data** to import data from a *.tab file.

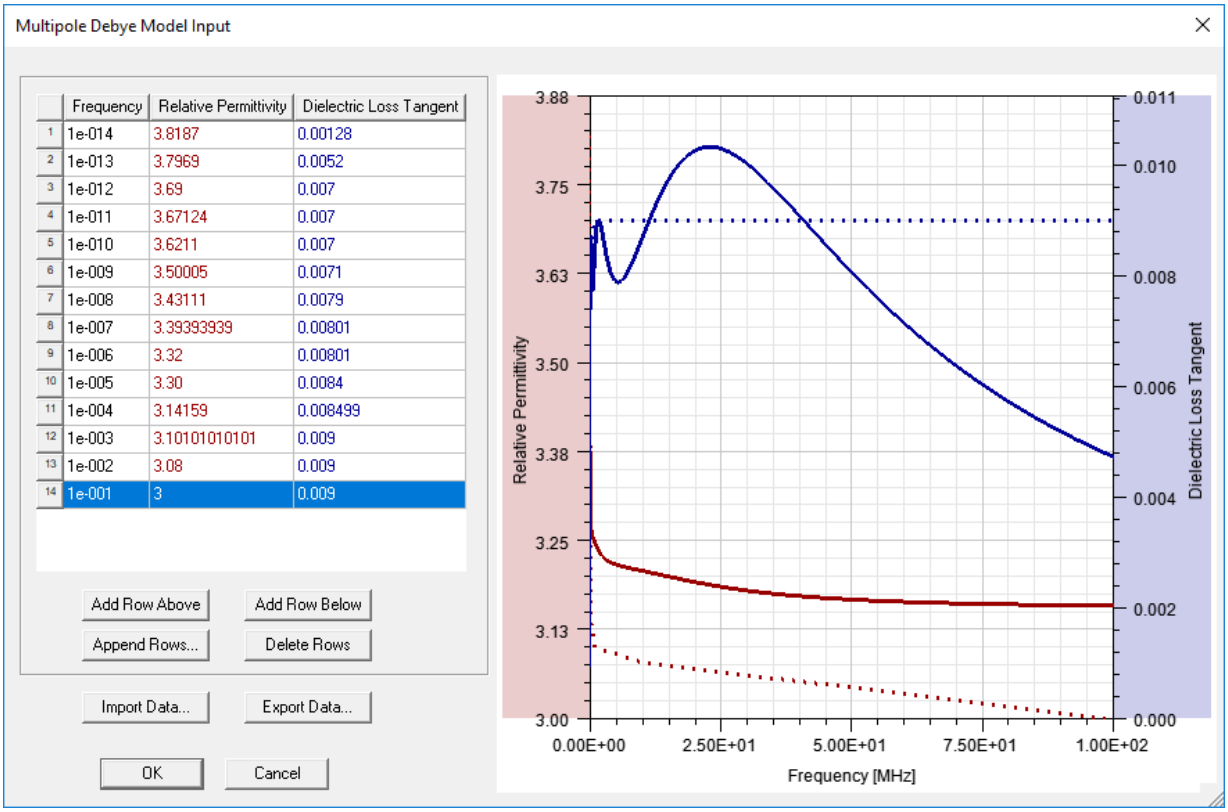
Below is an example of the file format. Each row provides Frequency (assumed to be Hz), Permittivity, and Loss Tangent.

```
0.00001 3.8136 0.00128

0.00010 3.7914 0.00520

0.00100 3.7500 0.00700
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0.01000 3.7119 0.00700
0.10000 3.6742 0.00700
1.00000 3.6354 0.00700
2.15444 3.6346 0.00702
3.17000 3.6325 0.01073
4.64160 3.6186 0.01500
10.0000 3.5777 0.01750
21.5444 3.5458 0.01750
26.0000 3.5383 0.01750
46.4160 3.5148 0.01750
50.0000 3.5119 0.01750
```

As you enter data, the plot updates. The input data is linearly interpolated using the Multipole Debye model.



Double-click any plot element to change its properties (color, units, scaling, etc.).

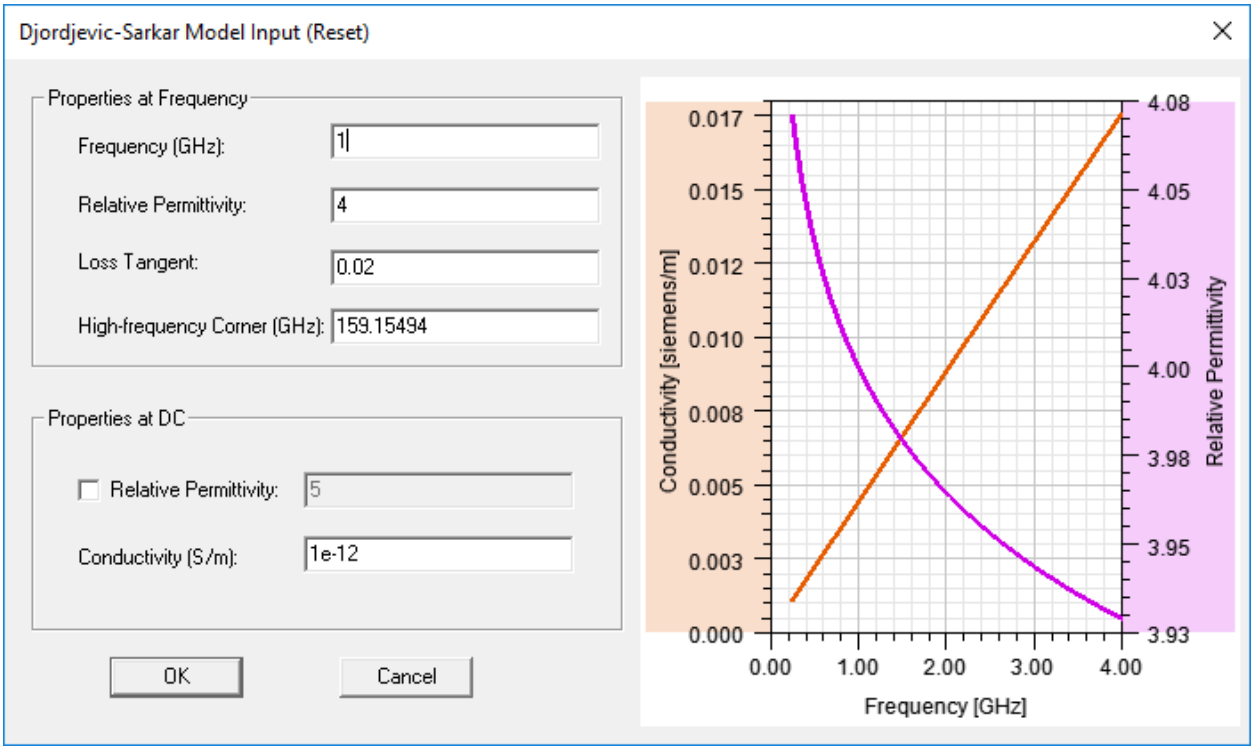
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To export the dataset in *.tab format for later use, click **Export Data**.

Assigning Frequency Dependent Material: Djordjevic-Sarkar Model Input

When you select **Djordjevic-Sarkar Model Input** as the model for the frequency dependent material property, the **Djordjevic-Sarkar Model Input** window appears.

From this window, you can enter material property values and view a property vs. frequency plot.



Use the fields to enter **Properties at Frequency** values:

- **Frequency**
- **Relative Permittivity**
- **Loss Tangent**
- **High-Frequency Corner** – the value of the High-frequency Corner should be at least 10 times higher than the Frequency. Reducing the upper corner frequency is not very attractive, because it has not been observed in experimental data. Therefore it is an upper bound on the loss tangents the Djordjevic-Sarkar model can handle. For technical details see *Technical Notes*: Djordjevic Sarkar Causal Dielectric Model.

Use the fields to enter **Properties at DC** values:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

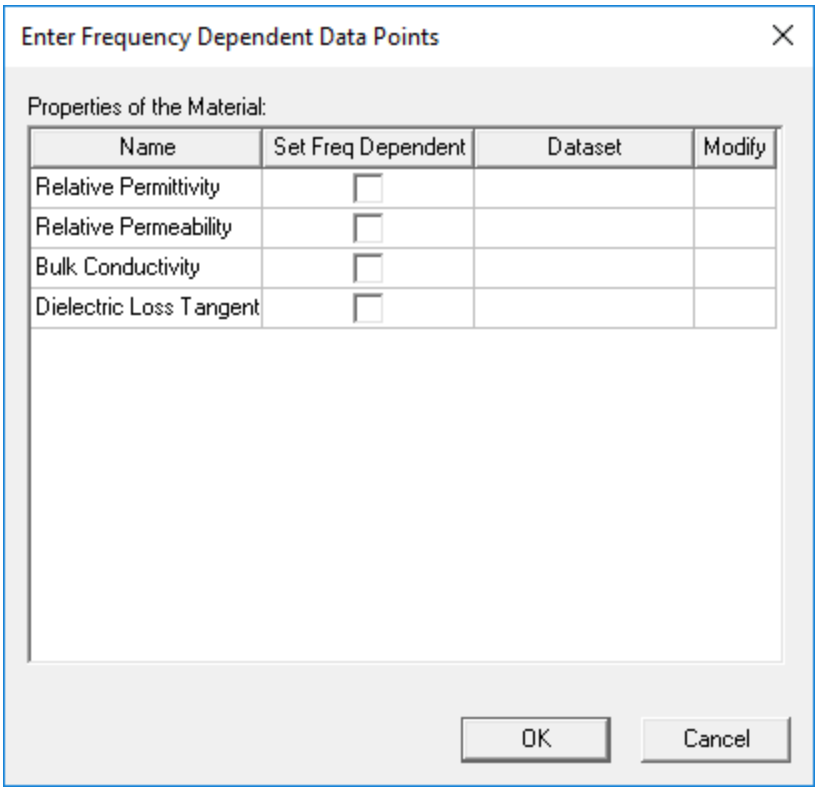
- **Relative Permittivity** – select the check box if you wish to specify relative permittivity.
- **Conductivity**

After you have entered the data for your selection, click **OK** to return to the **View/Edit Material** window. New default function names appear in the material property text boxes. A dataset is automatically created for each material property.

To modify the dataset with additional points, see Editing Datasets.

Assigning Frequency Dependent Material: Enter Frequency Dependent Data Points

When you select **Enter Frequency Dependent Data Points** as the model for the frequency dependent material property, the **Enter Frequency Dependent Data Points** window appears.

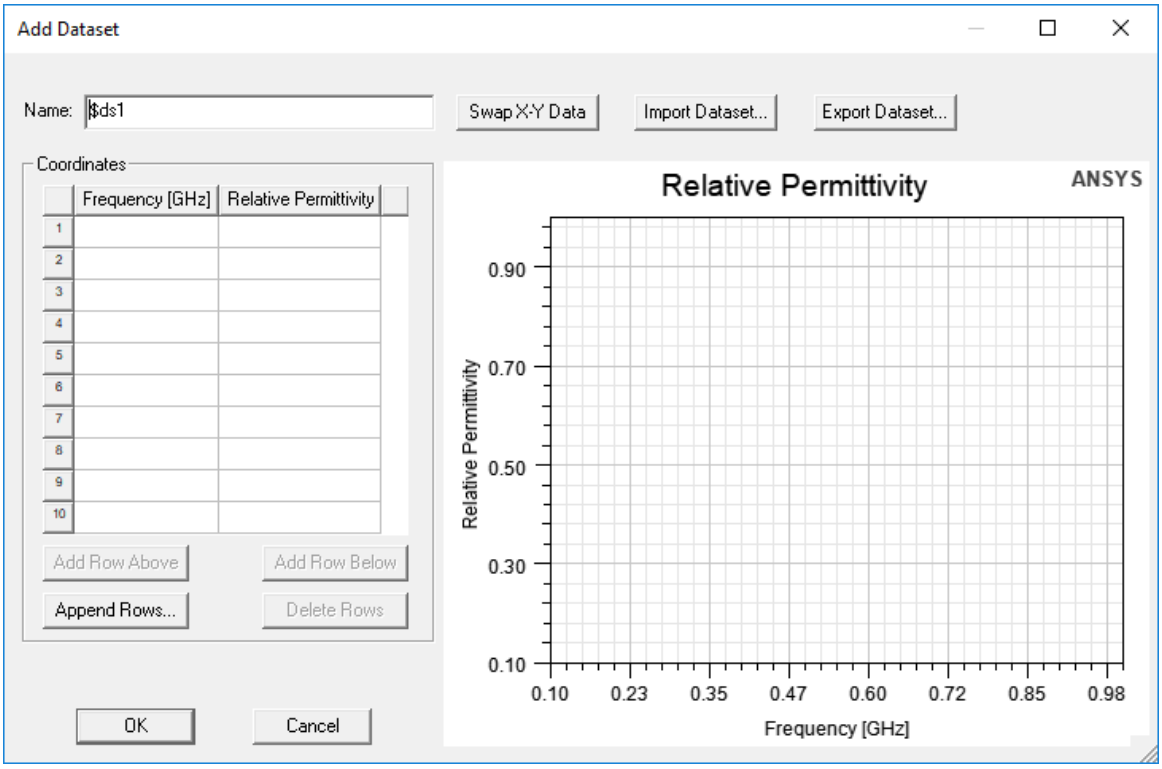


From this window, you can use datasets to set frequency-dependent material properties.

1. Select the **Set Freq Dependent** check box next to the property you wish to edit. This enables the **Dataset** field.
2. Click the property's **Dataset** field and use the drop-down menu to select **Add/Import 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.



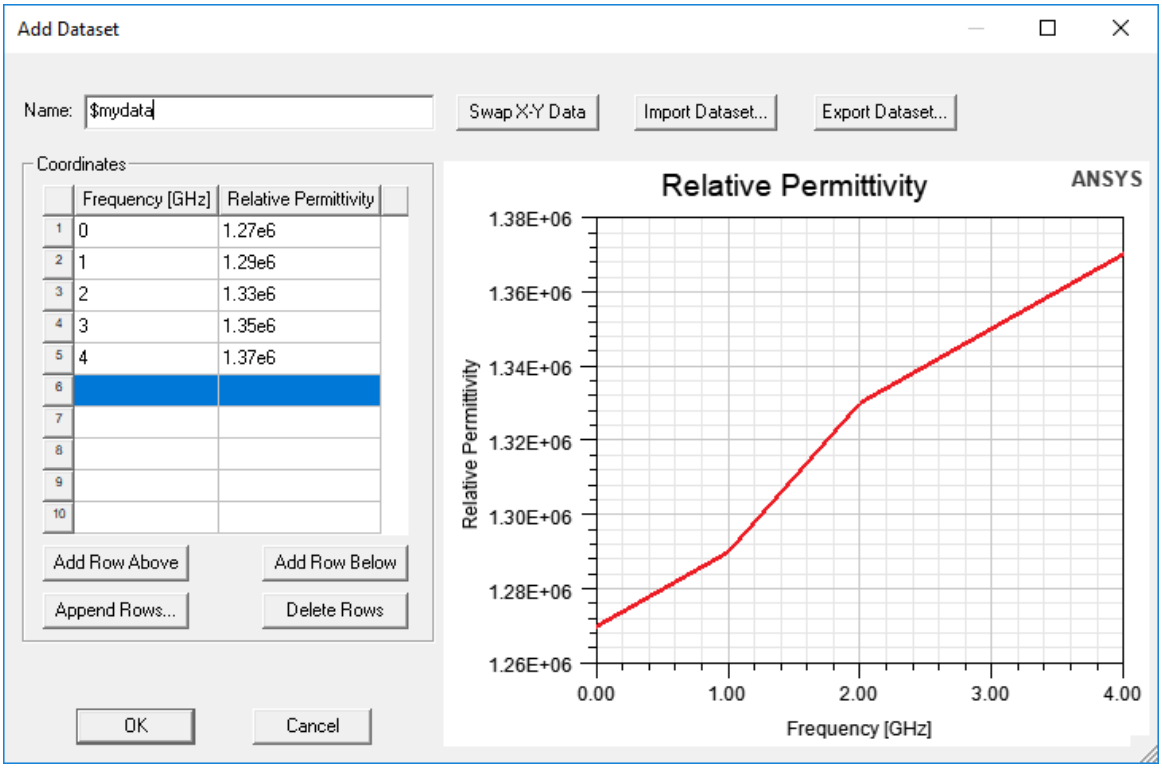
This window allows you to specify the following:

- **Name** – the default name is $ds1, but you can rename it. Note that the dollar sign ($) is automatically appended.
- **Coordinates** – contains a table that allows you to specify the Frequency (in GHz) and the specified material property's value at that frequency. Enter the data into the table manually, or click the **Import Dataset** button to populate it from a *.tab file.

You can also:

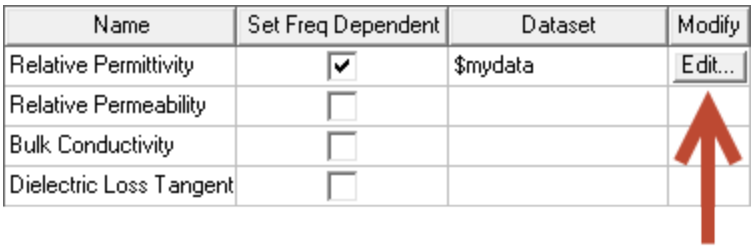
- Rearrange table items using the **Add Row Above**, **Add Row Below**, **Append Rows**, and **Delete Rows** buttons.
- **Swap X-Y Data**
- **Export Dataset** as a *.tab file for later import.
- View the table data as a plot on the right side of the window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. When you have finished adding points, click **OK**.

   You are returned to the **Enter Frequency Dependent Data Points** window, where the **Modify** field is now active. If you need to modify the dataset, click **Edit**.



4. Click **OK** again to return to the **View/Edit Material** window.

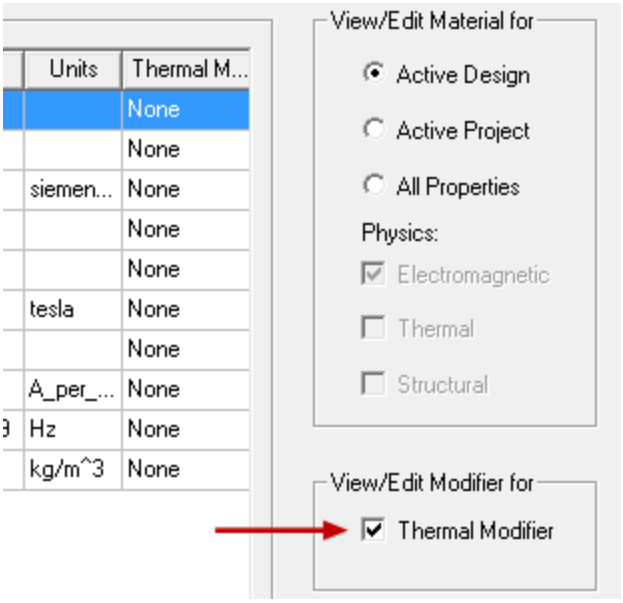
Specifying Thermal Modifiers

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.

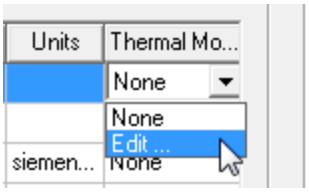
1. In the **View/Edit Material** window, you must enable the **Thermal Modifier** check box:



This option causes the properties table to expand to include a Thermal Modifier column.

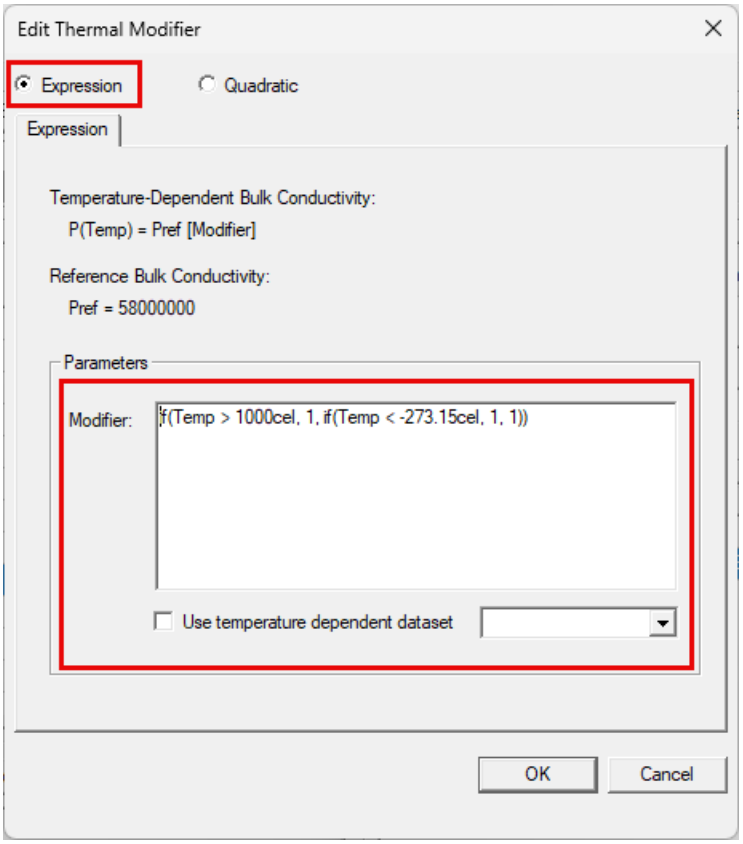
By default, the **Thermal Modifier** option is set to **None** for all material properties.

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.

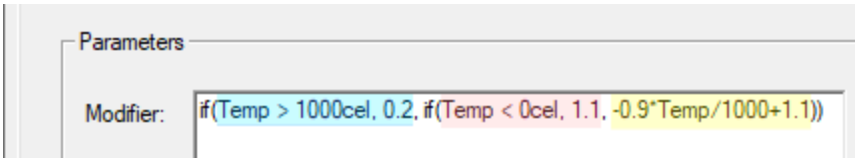
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:

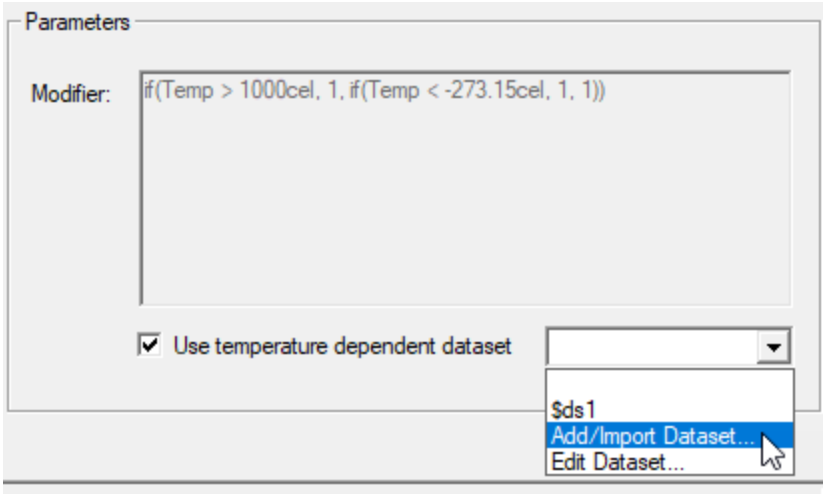


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, the linear equation **-0.9 * Temp / 1000 + 1.1** (highlighted in yellow) defines the multiplier for the range 0° C ≤ Temp ≤ 1000° C.

The default unit of temperature in thermal modifier expressions is Celsius (cel). However, you can override the units by appending the appropriate abbreviation to the numbers in the expression (mkel, ckel, dkel, kel, cel, rank, fah). Model temperatures, material properties, and thermal modifiers need not have consistent units; values are converted automatically as needed.

- Check **Use temperature dependent data set** to disable the Modifier text box. You can then use the drop-down menu at the bottom-right corner of the dialog box to select an existing project dataset or to create a new one by choosing **Add/Import Dataset**.

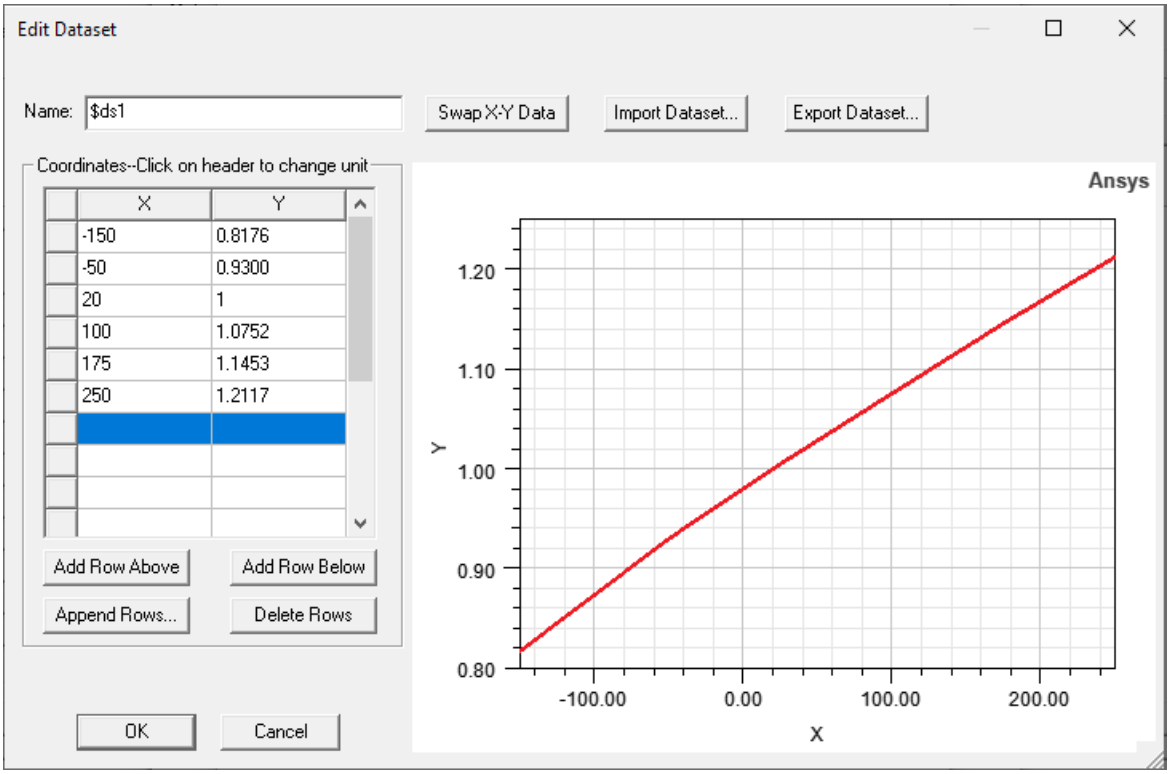


This command lets you define the thermal modifier dataset (a set of X and Y values). Alternatively, if a suitable project dataset already exists, it will be available for selection from the same drop-down menu.

When creating a new dataset, the **X** values are temperatures, and the **Y** values are the corresponding thermal modifiers. The value of the nominal material property is multiplied by this thermal modifier to determine the resultant property at a given temperature. The modifier is interpolated linearly between specified data points.

The following example is a dataset defining thermal multipliers for the thermal expansion coefficient (TEC) of an aluminum-beryllium alloy. The temperature range is -150 to 250° C, and the nominal value (corresponding to 20° C) is 1.37e-5/°C:

Ansys Electromagnetics Suite 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.

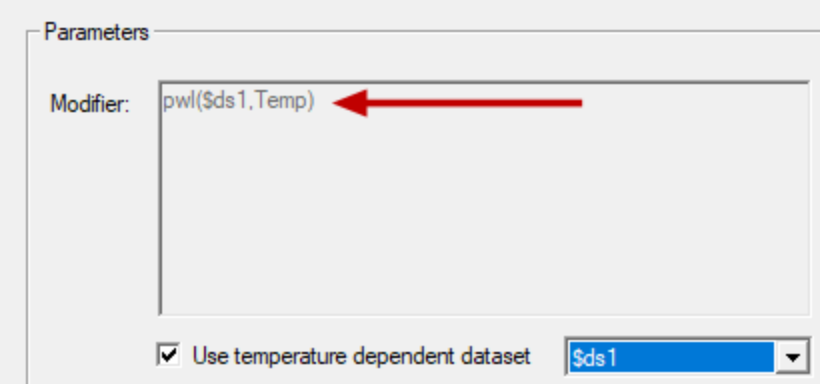
Alternatively, you can create a dataset based on any desired temperature units before defining your temperature-dependent material properties. To do so, click **Project** > **Datasets** from the menu bar and then choose **Add**. If your project includes a design type that supports 3D datasets, a submenu 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.

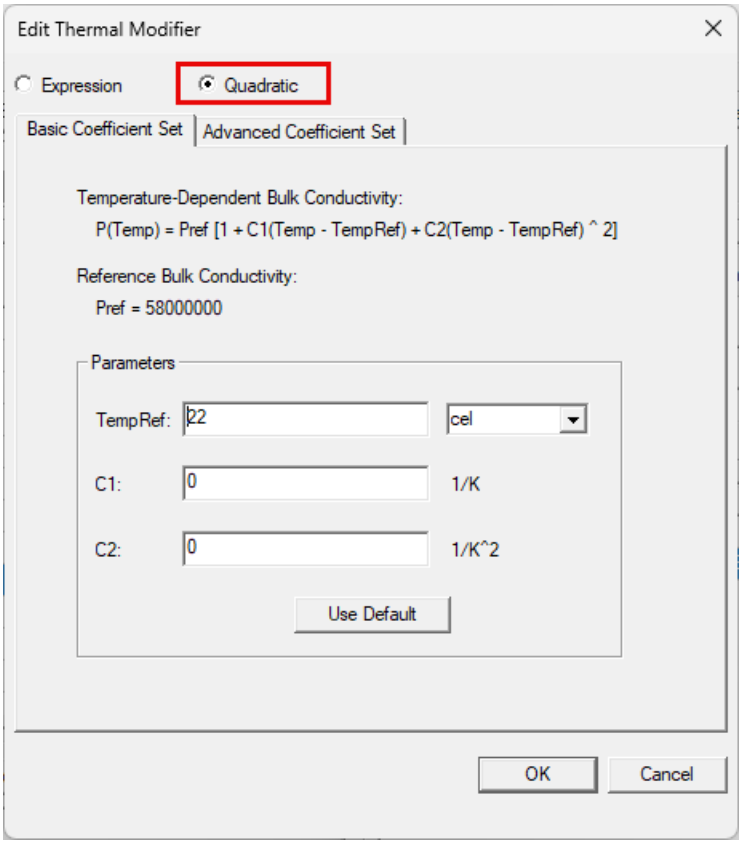
The syntax of the dataset expression is explained as follows:

- **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:

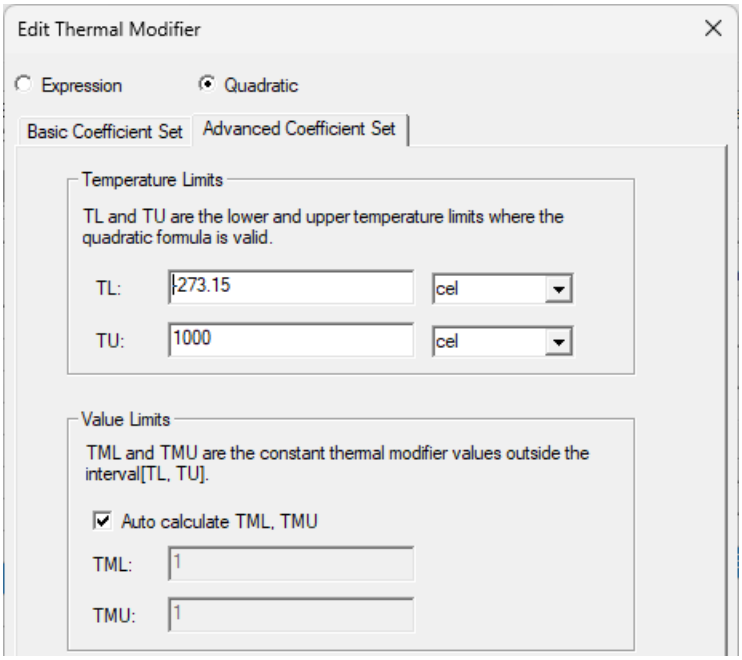
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

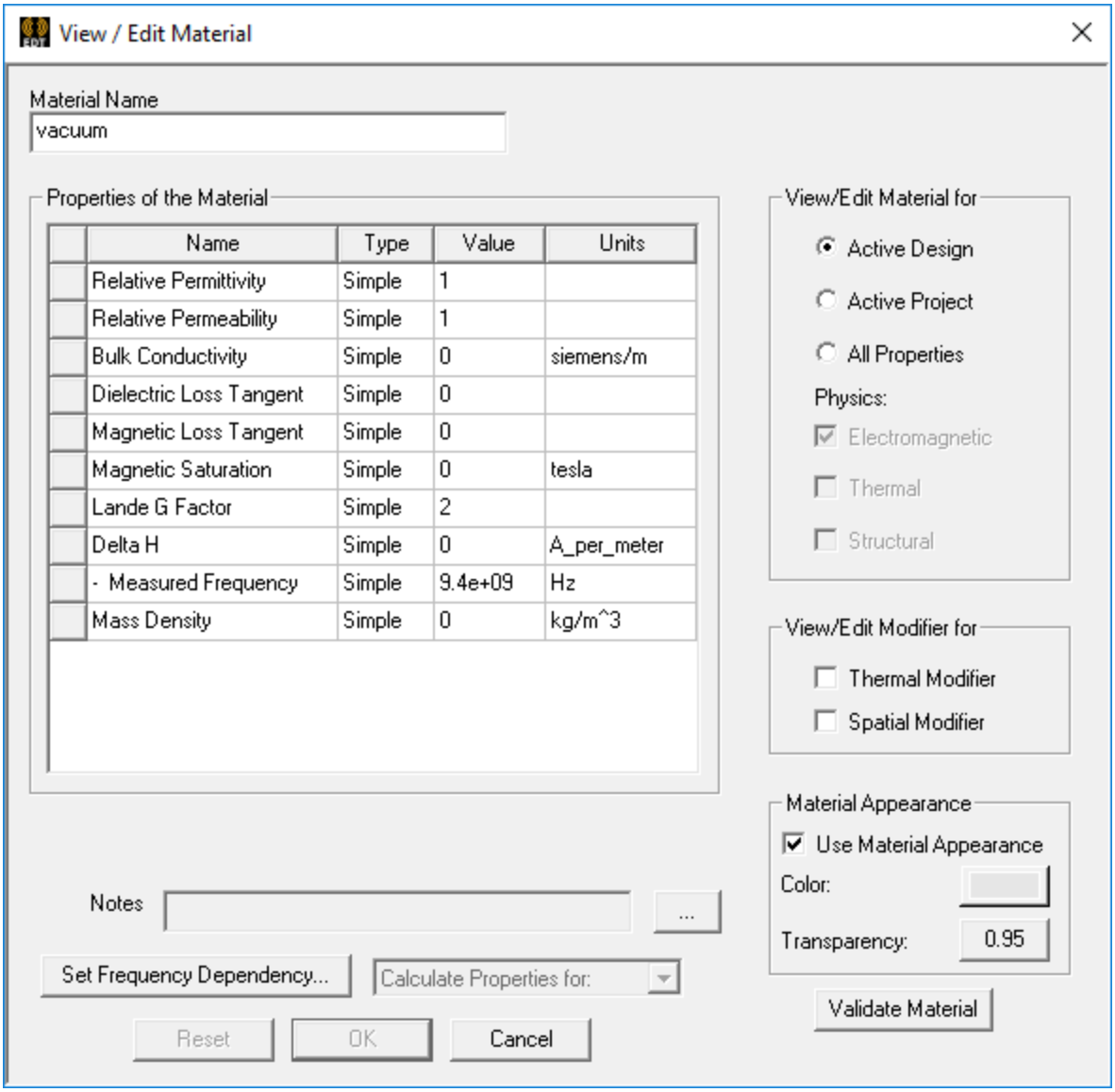
The Properties dialog for each object includes a materials property. If you click on the current material property you see a drop down list that includes an *Edit* command and a list of materials in the current project. You can select from the list of current materials to assign the selected material to that object.

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 design but physics type was not set), the **Physics** type will be automatically set in the material.

3. You can modify the material as follows:
   a. Provide a new name for the material in the **Material Name** text box.
   b. Depending on material property selected, the types available may be **Simple Anisotropic** as required for that property.

      For **Simple**, you provide a value or variable.

      For **Anisotropic**, you provide tensor values.

   c. Provide new material property values in the **Value** boxes.
   d. Change the units for a material property.
   e. Enable the **Use Material Appearance** check box to specify a color and transparency.

> **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.

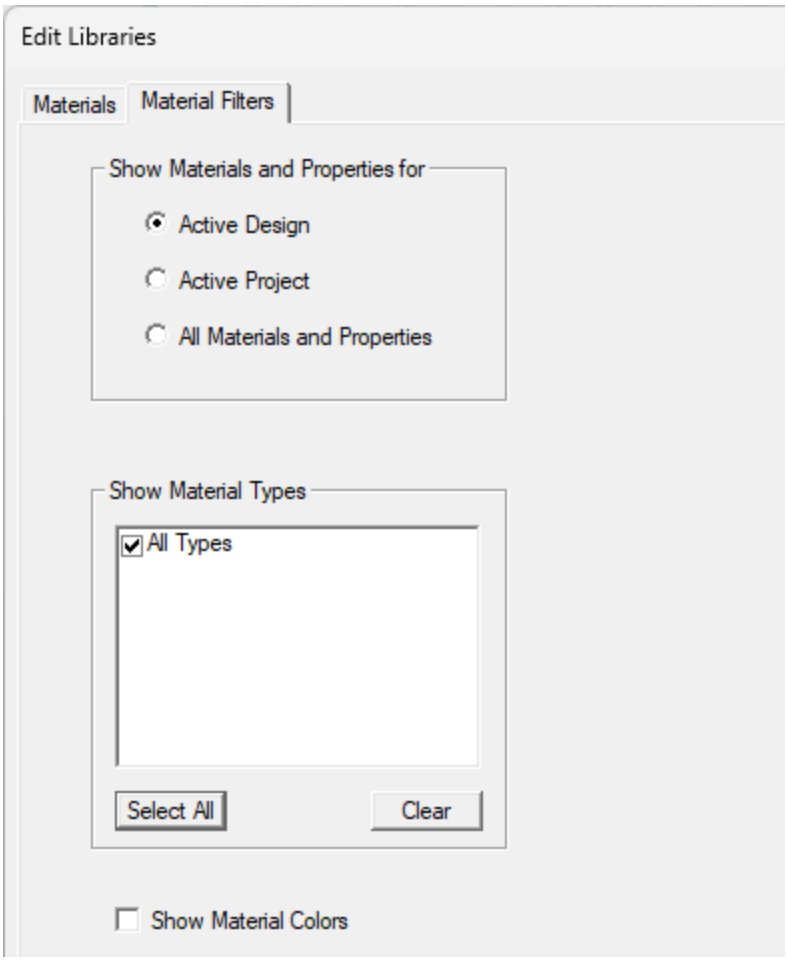
> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

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** tab. You can filter out materials based on the product or library with which they are associated. You can also filter out

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 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** tab:

1. Click the **Material Filters** tab.
2. Under **Show Materials and Properties for**, select one of the radio buttons:
- **Active Design** to display materials/properties used in the active design.
- **Active Project** to display materials/properties used by the active project.
- **All Materials and Properties** to display all materials and properties available. Selecting this enlarges the table of materials shown under the **Materials** tab to show all materials possible. You can use the scroll bars or size the dialog to see all materials.
3. Under **Show Material Types**, select materials types to display on the **Materials** tab. Click **Select All** to select all of the types listed. Click **Clear** to clear all selections.
4. If you check Show Material colors, the Materials tab of the **Select Definition** window includes a color for material colors. See Viewing and Modifying Material Attributes.
5. Click the **Materials** tab to save your selections.

Click **Cancel** to revert back to the last saved selections.

Working with Material Libraries

There are three different materials libraries in HFSS 3D Layout: 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.

9 - Port Setup in HFSS 3D Layout

When you specify an excitation for an HFSS 3D Layout structure, HFSS 3D Layout uses the principle of linear superposition to calculate the solution. The signal on each port and mode is solved for separately. HFSS 3D Layout sets the signals on all non-zero modes and generates a solution. This procedure is repeated with each port until a solution is generated for each.

HFSS 3D Layout offers a variety of ports. HFSS attributes and sizing information are all available as properties. This includes port sizes, selected return paths, and verifying port types as being selected for the particular application that is being designed and simulated.

HFSS 3D Layout automates the following processes to ensure the successful setup of ports:

- Automatic reference location (with override option)
- Automatic port-geometry generation (with sizing override option)
- Automatic terminal identification
- Numerous pre-solve validation checks, with messaging to alert you when something is wrong

The following topics describe the setup and usage of the various ports available in HFSS 3D Layout.

HFSS 3D Layout Port Types

The HFSS port types available in HFSS support configurations that are common for the excitation of most layered designs.

- Wave ports are available to excite common transmission-line cross-sections.
- Lumped ports are available to excite internal structures or transmission lines, and can be vertical, horizontal, or coaxial.
- Deembedded lumped ports in HFSS 3D Layout are not restricted to being rectangular, they can be arbitrary in shape.
- A parallel excitation exists for the HFSS 3D Layout so that two solvers are available for one design setup.

The following HFSS port types are available in HFSS.

- Ports for Transmission Lines (Edge Ports or Vertical Lumped Ports)
- Ports for Imported Package and PCB Structures
- Creating Pin-Based Ports
- Configuring Port-Instance Ports
- Gap(coax) Ports

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Wave(coax) Ports
- Horizontal Wave Ports
- Horizontal Lumped Ports
- CPW and Slotline Ports
- Circuit Ports and Elements

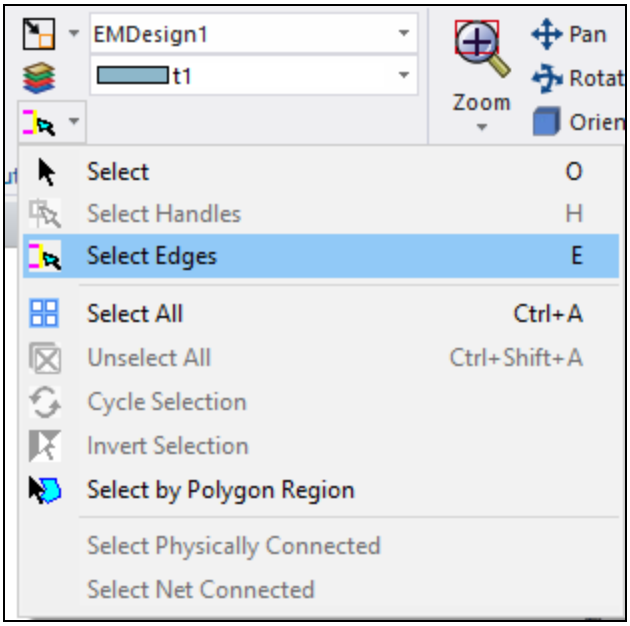
Ports for Transmission Lines

HFSS ports for transmission lines can be set up as single-line or multi-line edge ports. Edge ports (i.e., vertical lumped ports) can be added to any primitive. By default, the edge ports are port boundaries. However, you can make boundaries that are perfect electrical conductors (PEC) or RLCs, or change an existing edge port to a PEC or RLC. Edge ports are listed under **Excitations** in the **Project Manager** window. Boundaries are listed under **Boundaries**.

Creating Edge Ports

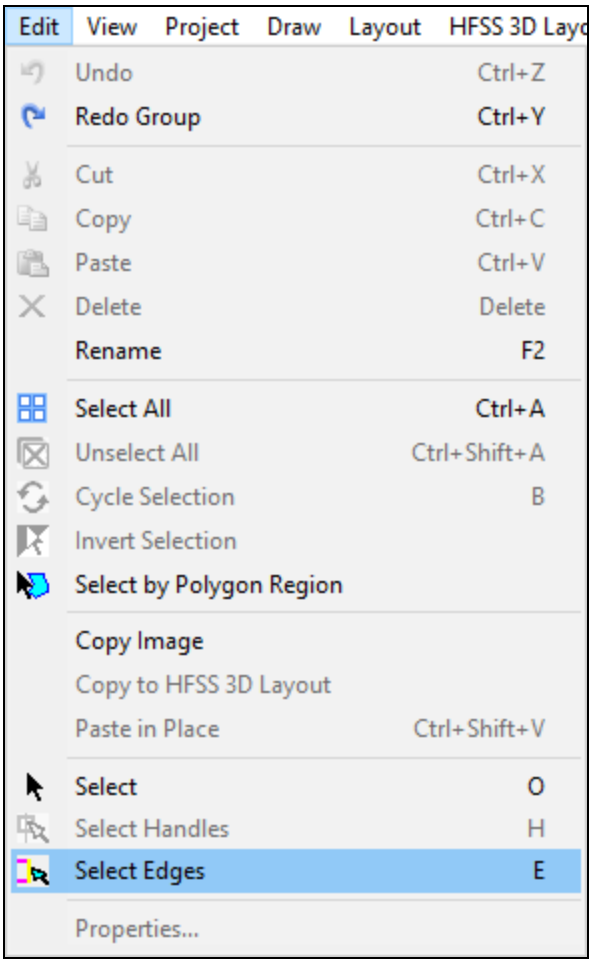
Follow these steps to create an edge port.

1. To begin, do any of the following:
   - Press **E** to enter **Select Edges** mode.
   - From the **Layout** tab, click **Select edges** > **Select Edges**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From **Edit**, click **Select Edges**.

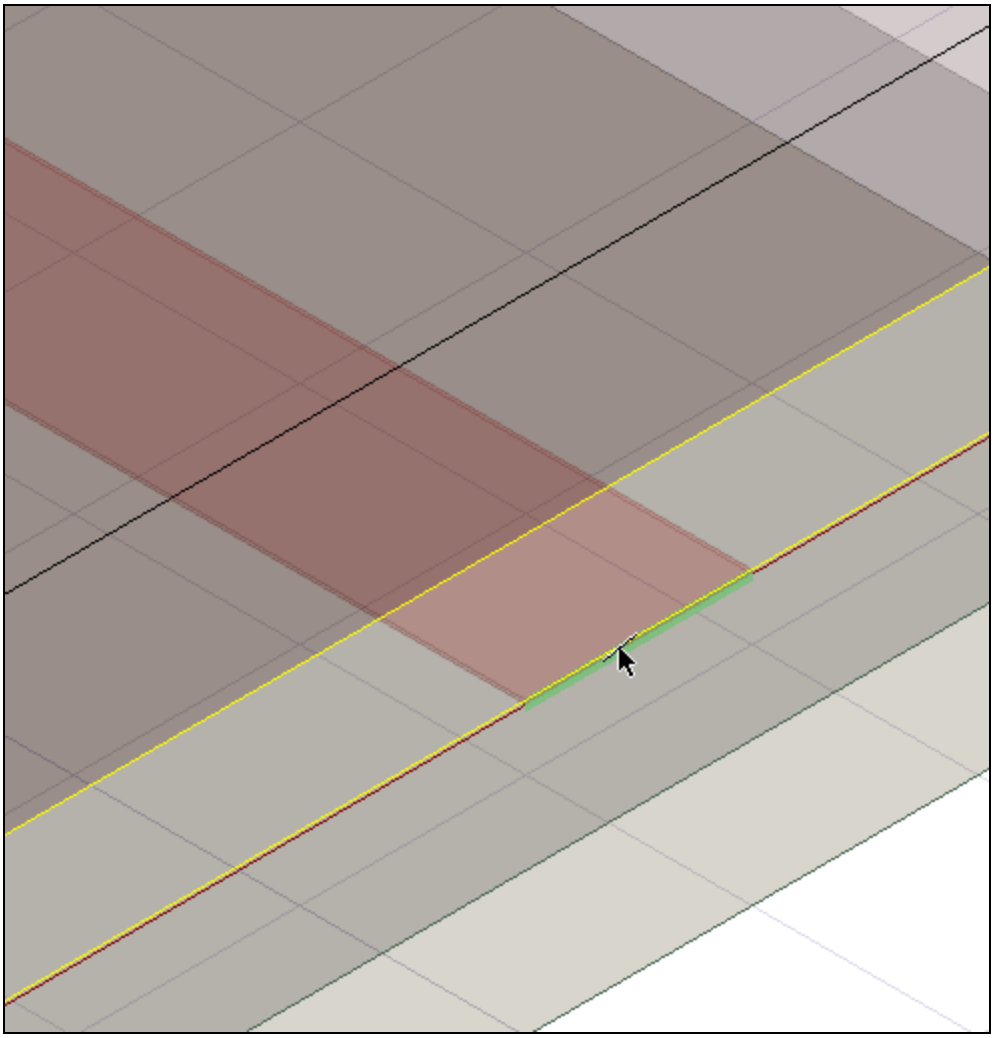


Note:

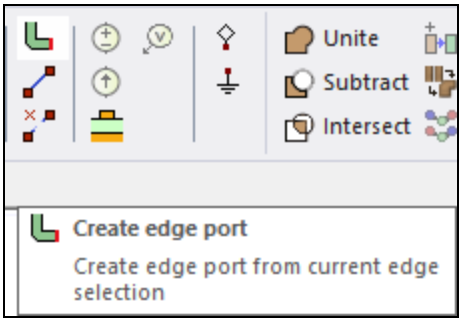
Once **Select Edges** is selected, the cursor changes: a diagonal line crosses the tip of the arrow. Refer to the following step.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the primitive at the appropriate location to add a port.

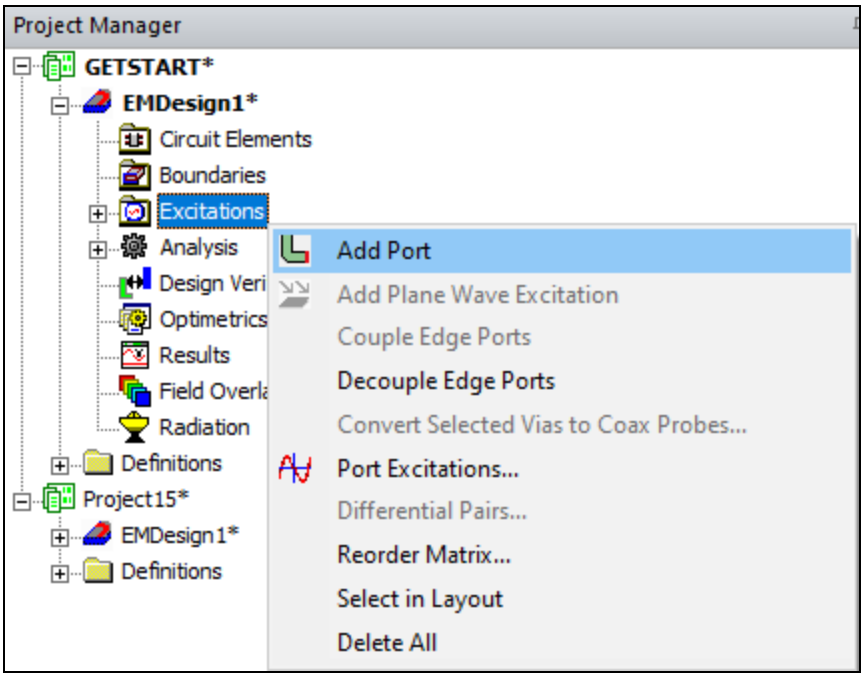


3. To create the edge port, do one of the following:
   - From the **Layout** tab, select **Create edge port**.



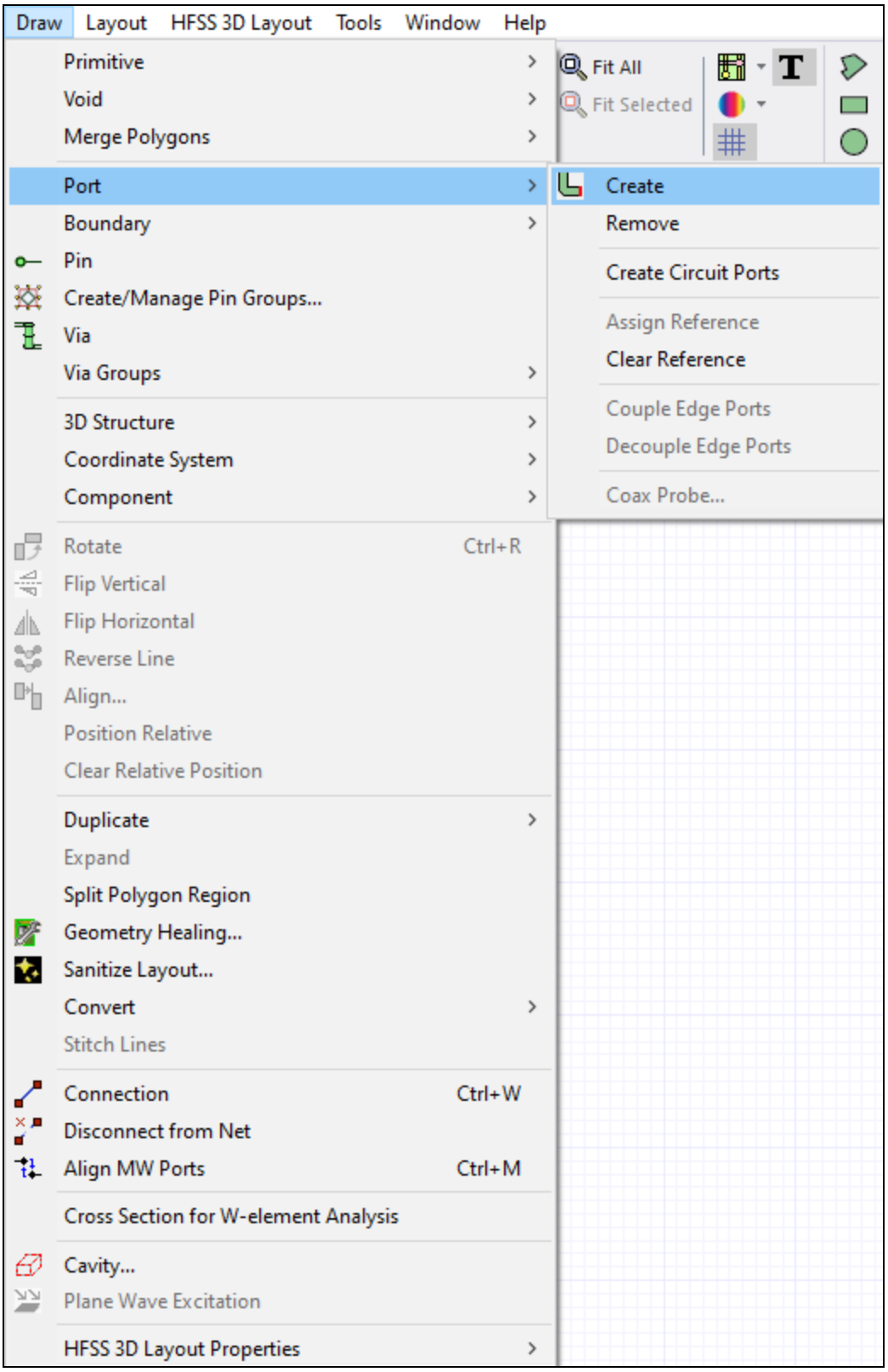
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the **Project Manager** window, expand the **Project Tree** and ***active design folder*** **(e.g., EMDesign1**. Then right-click **Excitations** and select **Add Port**.



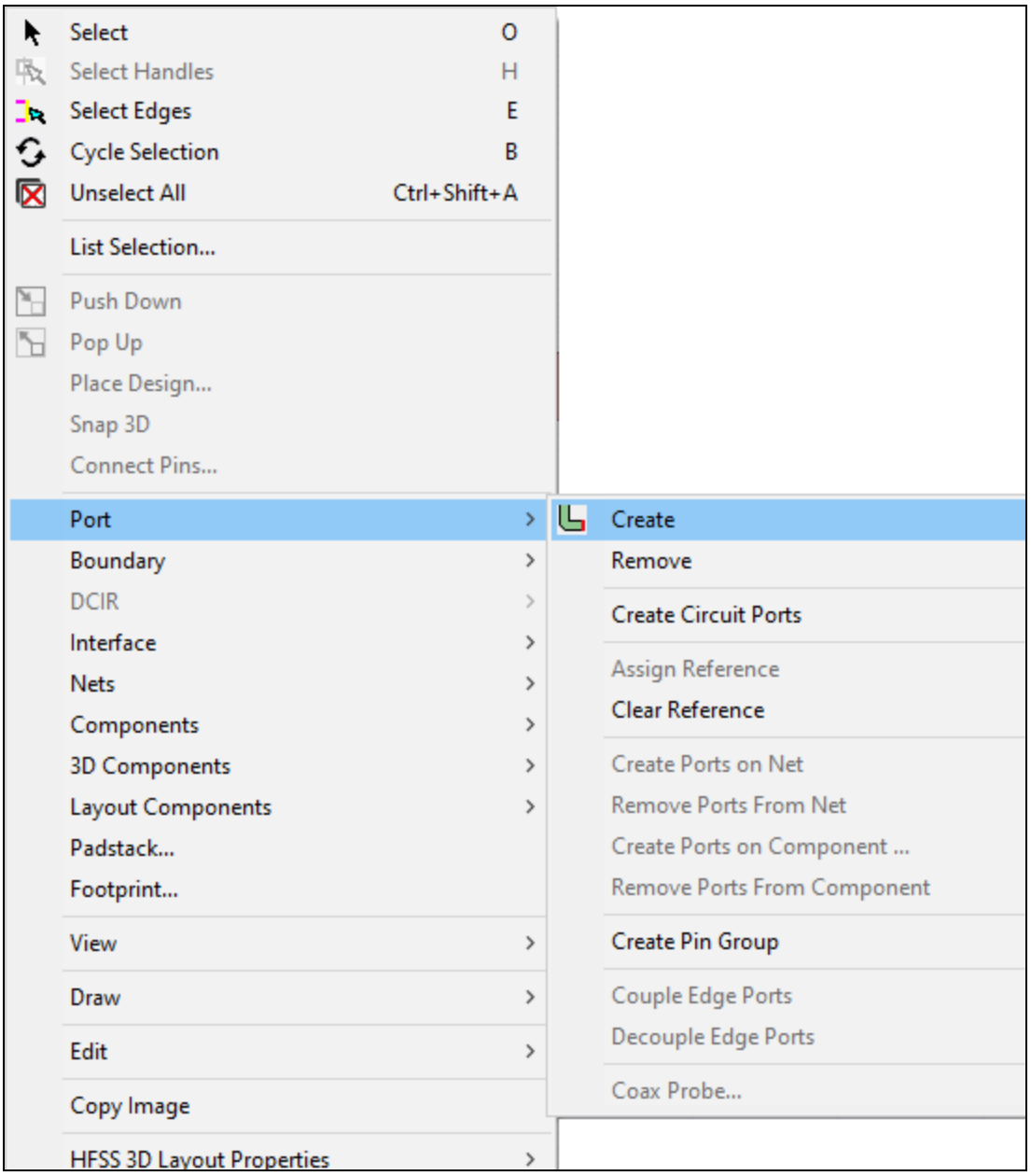
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From **Draw**, select **Port** > **Create**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

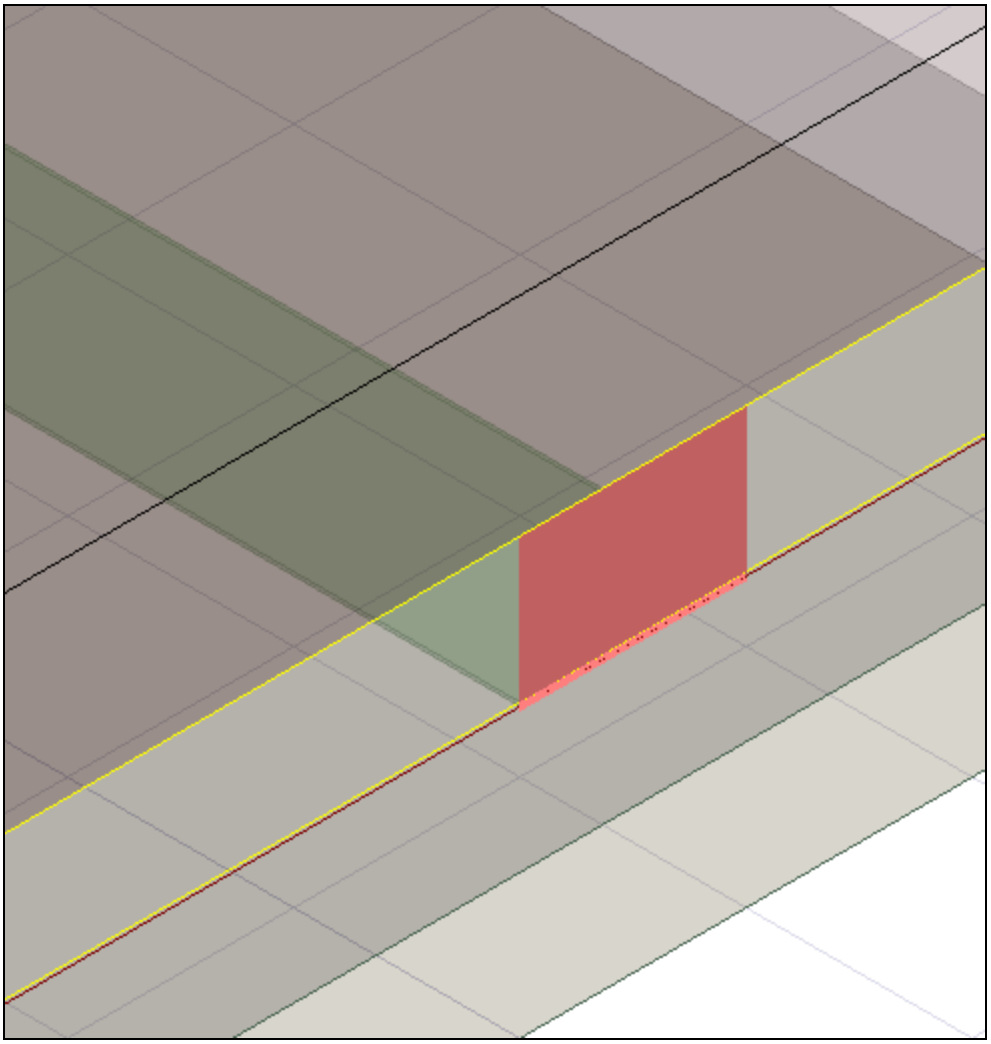
- Right-click in the **Layout Editor** and select **Port** > **Create**.



> **Note:**
>
> Once a port is created, it appears in the **Project Manager** > **Project Tree** > **[*active design folder*]** > **Excitations** folder.

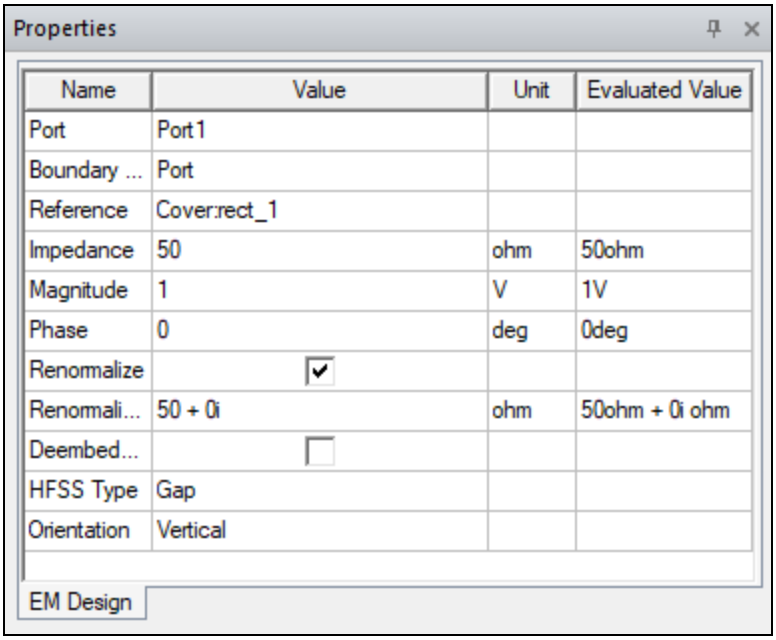
The port appears in the **Layout Editor**.

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 port is selected, the **Properties** window populates with its 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 **Reference** surface automatically defaults to the closest signal layer (with geometry) or negative layer, above or below the primary terminal. If the port abuts two possible reference surfaces (i.e., two planes), users can select either surface or both (i.e., dual reference surfaces).

Creating PEC Edge Ports

1. From the Layout Editor, select an edge of the primitive that has the edge port.
2. Either:
   - Click **Draw** > **Boundary** > **Create PEC**.
   - Right-click and select **Boundary** > **Create PEC**.

Creating RLC Edge Ports

1. From the Layout Editor, select an edge of the primitive that has the edge port.
2. Either
   - Click **Draw** > **Boundary** > **Create RLC**.
   - Right-click and select **Boundary** > **Create RLC**.
3. From the **Properties** window, open the **EM Design** tab.
4. Set the resistance, capacitance and inductance values in the **R**, **L**, and **C** rows.

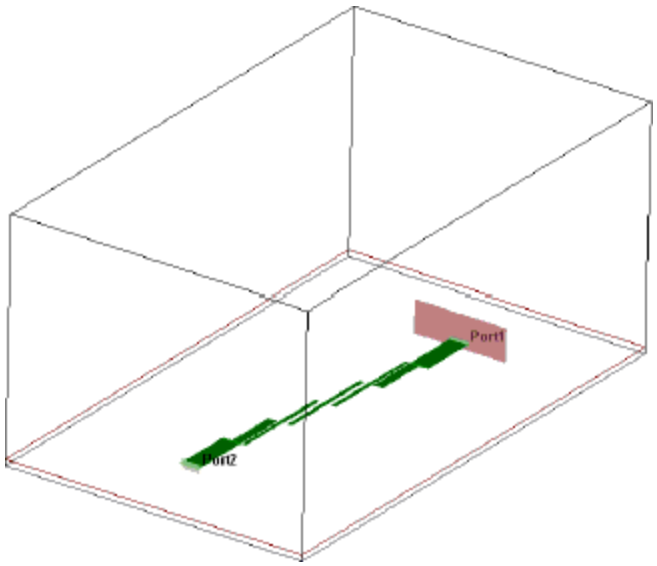
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the Type of Port or Boundary

1. From the Layout Editor, select the port or boundary.
2. From the **Properties** window, open the **EM Design** tab.
3. From the **Boundary Type** row, select the new type.

Single-Line Transmission Line Ports

When configuring HFSS ports as single-line transmission ports, the edge selections must be a continuous, co-linear set of segments.



To convert between a lumped-gap and a wave port, toggle the HFSS Type Attribute in the HFSS 3D Layout Properties. The 3D view are updated automatically. Additional properties that can be set include:

General properties

- Reference -select geometry/layer for vertical constraint
- Impedance -if Z0 cannot be determined from port solution
- Post-processor –renomalization and deembedding

Gap

- Orientation (read only)

Wave

- Orientation (read only)
- Horizontal Extent Factor –if applicable, sizing parameter for vertical wave ports

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- PEC Launch Width –dimension for port-geometry thickness.

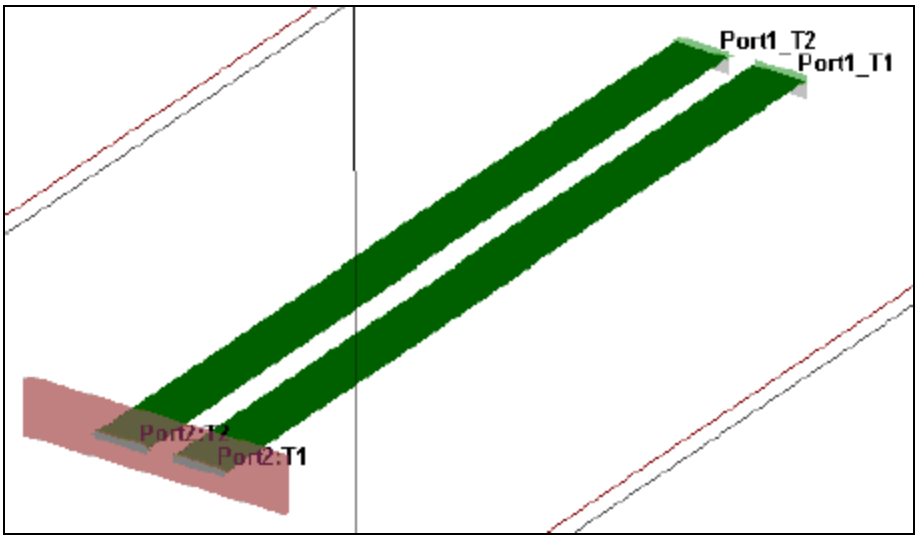
The following pre-simulation validation checks apply:

- Port geometry may not intersect other port geometry
- Port geometry may not intersect layout geometry
- Wave ports may not contain “floating” terminal geometry

Multi-Line Transmission Line Ports

When configuring HFSS ports as multi-line transmission ports, wave ports are created by default, but can be “decoupled” to create a collection of lumped-gap ports. To decouple a multi-line coupled port:

1. Select the port graphically, or click **Layout** > **List**, then select the port
2. Click **Draw** > **Decouple Edge Ports**



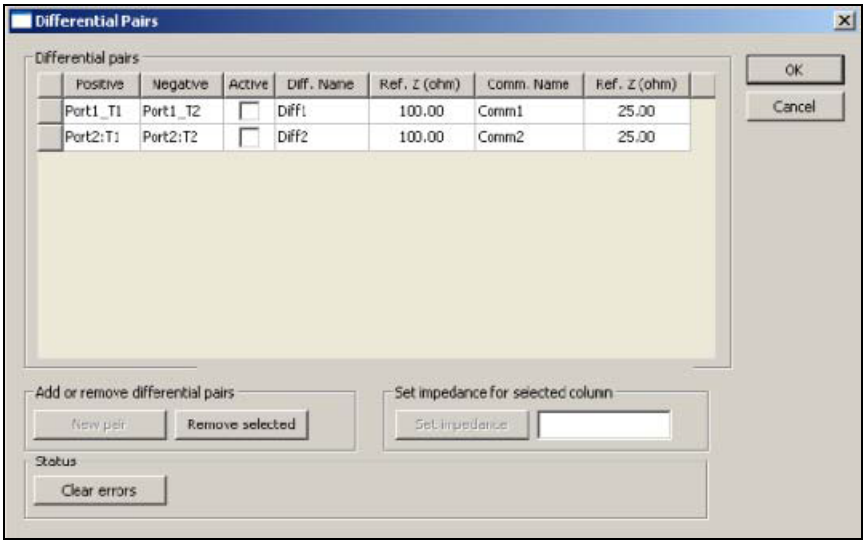
Similarly, ports can be coupled using **Draw** > **Couple Edge Ports**. Edge selection must be co-linear, but can span multiple planes.

The following pre-simulation validation checks apply:

- Geometry intersections
- Floating terminals

Solution are always terminal, but differential-pairs may be configured separately using the **Differential Pairs** 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 Coupled Ports From Existing Edge Ports

To create coupled port from an existing edge port:

1. Create two or more edge ports using the procedure described in Ports for Transmission Lines.
2. Click the **Select elements** tool to select the edge ports that you want to include in the layout editor. (Do not click the **Select Edges** tool.) The ports must lie in the same plane, be collinear and cannot touch.
3. Right-click **Excitations** in the **Project Tree** (or open the **Draw** menu) and select **CoupleEdgePorts**. The coupled ports now share the same port name, but have different terminal names: <PortName>:<TerminalName>.

To decouple ports, click one or more of the coupled ports and right-click **Excitations** in the **Project Tree** (or open the **Draw** menu) and select **Decouple Edge Ports**. Alternately, you can also decouple by selecting a port beneath **Excitations** in the project tree and selecting **Decouple Edge Port**. Also decouple by deleting one or more of the coupled ports.

Removing an Edge Port or Boundary

1. Select the edge port or boundary.
2. Do one of the following:
   - Click **Draw** > **Boundary** > **Remove**.
   - Click **Draw** > **Port** > **Remove**.
   - Right-click and select **Boundary** > **Remove**.
   - Right-click and select **Port** > **Remove**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ports for Imported Package and PCB Structures

From a geometrical perspective, a pin is nothing more than a via. However, pins must also provide a logical interface (a port) to the design in the schematic view. This means that all pins must publish an external port and the associated geometry for HFSS analysis.

Pins are crucial in HFSS because they are a fundamental element in most layout data models, so any translated designs likely includes them. While it is possible to interactively assign ports based on edges and primitive geometry, this approach scales poorly, particularly when pins already exist. Also, in HFSS, pins can be toggled to create internal PEC or RLC boundary conditions just like HFSS-3D.

Creating Pin-Based Ports

To create a new pin:

1. Click **Draw** > **Pin** and place the pin at the selected location in the Layout Editor.
2. From the **Footprint** tab of the **Properties** window, set the **Padstack Definition** or **Padstack Usage**.
    - Click **Padstack Definition** to open the Padstack definition library in the Select Definition window, where select a pre-defined padstack.
    - Click **Padstack Usage** to open the Padstack Usage and Definition editor where you can modify the padstack used by this pin.
3. Select the terminal **Location** by setting the **Pad port layer**. The **Pad port layer** selection is not displayed if the padstack has no pads. Click **Padstack Usage** to adjust the padstack if selected.

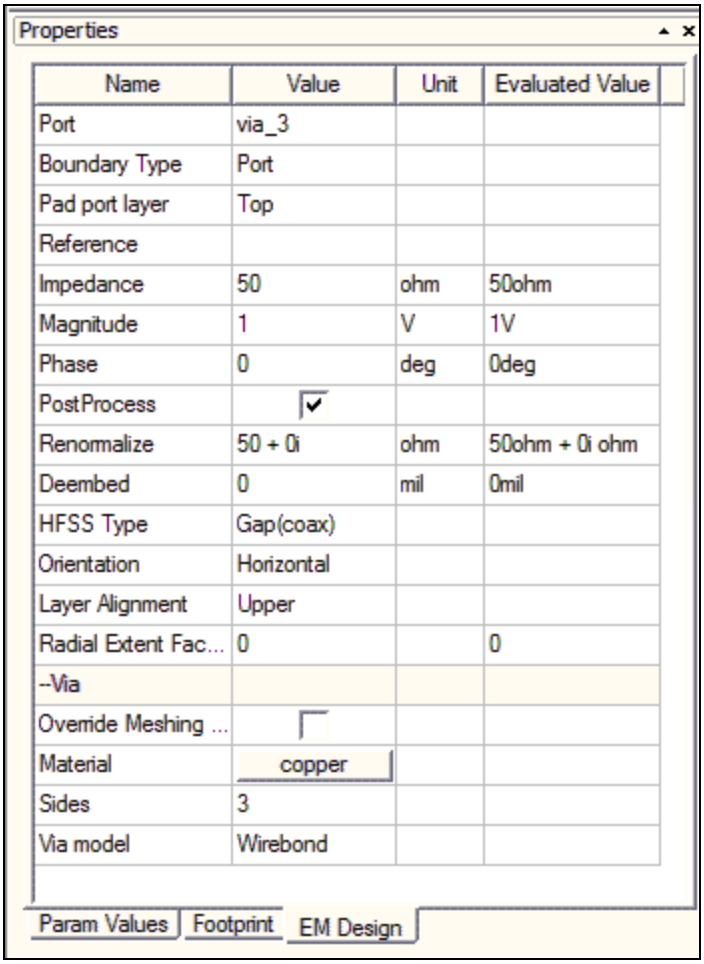
Configuring Pin-Based Ports

The following settings on the **EM Design** tab in the **Properties** window are HFSS 3D Layout properties. They apply to pin-based ports.

- **Pad port layer**: Sets the terminal location and pad-geometry used for derivation of the HFSS port.
- **Reference**: If applicable, select the geometry or layer for vertical constraint.
- **HFSS Type**: Either **Gap** or **Gap(coax)**.
- **Orientation** (read only)
- **Layer Alignment**: If applicable, select layer surface for port-geometry.
- **Radial Extent Factor**: Sizing parameter for port-geometry radius. Only appears if HFSS Type is **Gap(coax)**.

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 port-geometry radius is calculated as follows: Outer Radius = Inner Radius + (Radial Extent Factor) * Inner Radius. You can also specify a value with units. In that case, Outer Radius = Inner Radius + Radial Extent Factor with Units. When 0 is specified, the factor is set to 0.5 for solderballs and -0.2 for regular non-solderball pads.



Creating a Pad Port

1. Select the signal layer on which the pad port lies.
2. From the **Edit** menu, select **Select Edges**.
3. Click the edges of the model on which the port lies. The edges do not need to be collinear.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. From the **Draw** menu, select **Port** > **Create**. The new pad port is listed under **Excitations** in the **Project Manager** window.

HFSS calculates a characteristic impedance of 50 ohms for the pad port. The propagation constant is set to the propagation constant in Teflon, in order to agree with the propagation constant of coaxial probe ports.

To edit the pad port's properties, double-click the pad port in the **Project Manager** window. The **Edge Port Definition** window opens where you can perform these tasks:

- Change the name of the pad port
- Deembedding S-Matrices
- Vary the current excitation
- Renormalize the port to a specific port impedance
- Setup a Reference to the Nearest Negative Signal

When the pad port is selected, its **EM Design** tab is available in the **Properties** window. This tab displays many of the values in the **Edge Port Definition** window, including the port classification for the HFSS-PI type.

> **Note:**
>
> The desktop automatically defines all ports that connect N-ports to primitive objects as pad ports.

Configuring Ports

Using the following shortcut menu port commands, you can configure ports in the **Layout Editor**. Right-click within the **Layout Editor** and navigate to **Port** to view the following options.

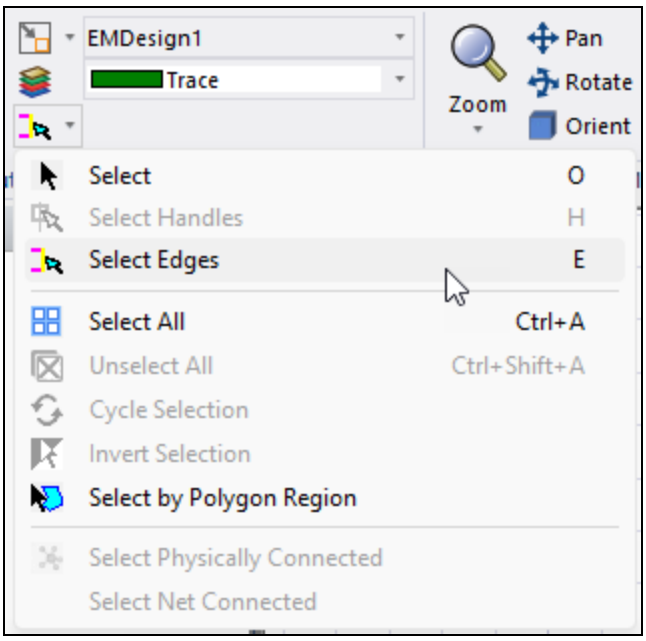
> **Note:**
>
> All Port options require the user to select one or more objects in the **Layout Editor**. These instructions assume no object has been selected and the shortcut menu is not open.

> **Note:**
>
> The screenshots in this section are illustrative only. Elements shown in the **Layout Editor** are intended to demonstrate cursor behavior and workflow steps, not valid configurations. These examples focus on tool usage, not electrical correctness.

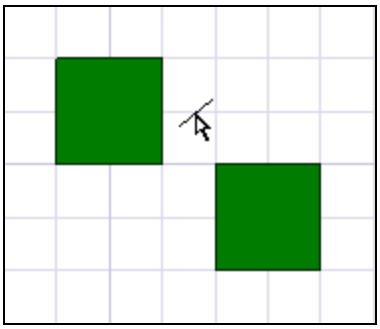
Create an Edge Port

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Do either of the following to enter edge selection mode:
   - From the **Layout Ribbon**, choose the **Select Edges** cursor.

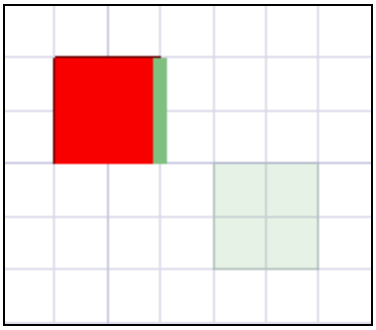


   - Enter **E** in the **Layout Editor** and wait for the cursor to transform.

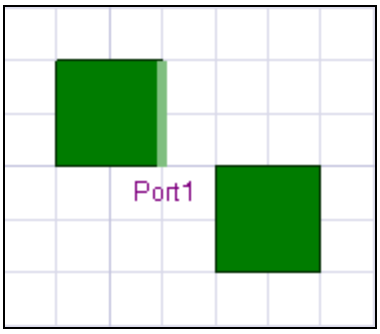


2. Select the edge of a primitive.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Right-click in the **Layout Editor** and select **Port** > Create to create an edge port.



> **Note:**
>
> Refer to Ports for Transmission Lines.

Remove an Edge Port

Right-click on an edge port and select **Port** > **Remove**.

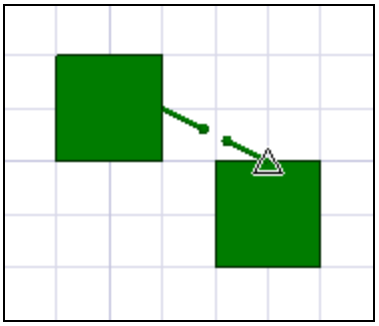
Create Circuit Ports

To transform an edge port into a circuit port, right-click on an edge port and select **Port** > **Create Circuit Ports**.

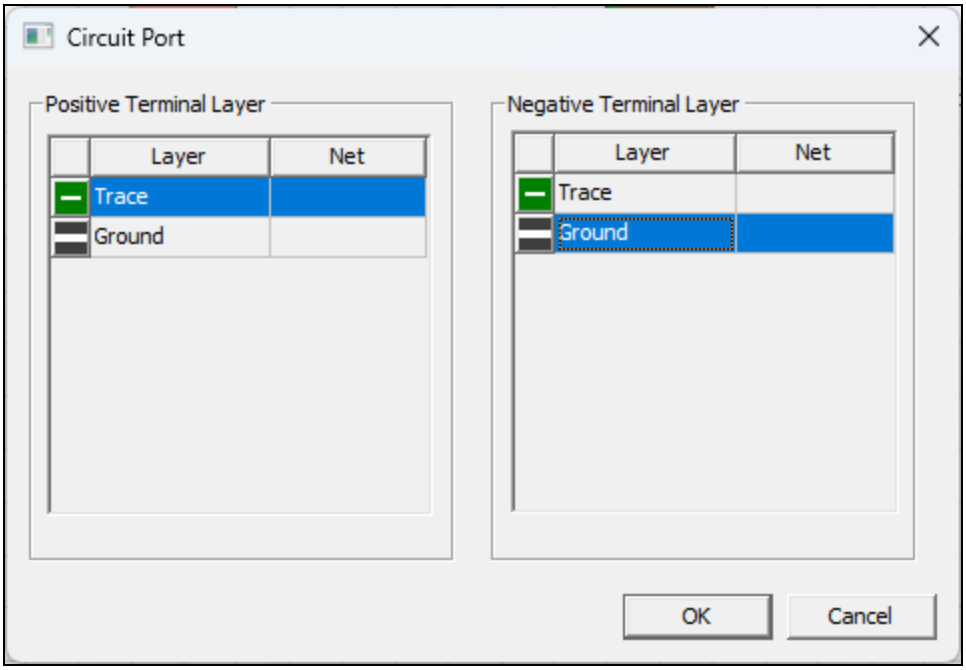
To create a circuit port:

1. Right-click in the **Layout Editor** and select **Port** > **Create Circuit Ports**.
2. Click the edges of two primitives to select the positive (i.e., first selection) and negative (i.e., second selection) terminal locations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

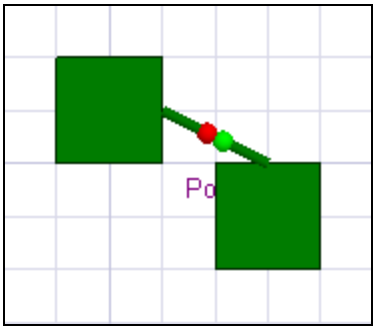


The **Circuit Port** window opens.



3. Select a **Positive Terminal Layer** and a **Negative Terminal Layer**.
4. Click **OK**.

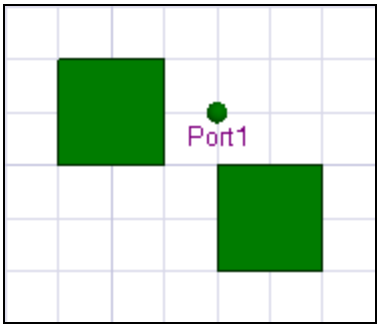
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create Terminal Ports

> **Note:**
>
> Terminal ports and circuit ports are functionally equivalent when solved by an HFSS solution setup. Both define point source excitations for simulation with the only distinction being that the reference terminal for a terminal port is automatically assigned. The distinction for RaptorX or Q3D solvers is different (e.g., the HFSS solver treats terminal ports as circuit ports with reference terminals automatically assigned, RaptorX and Q3D treat terminal ports as single terminal excitations without reference terminals). The **Layout Editor** provides separate creation options, but the solver decides how terminal ports are interpreted.

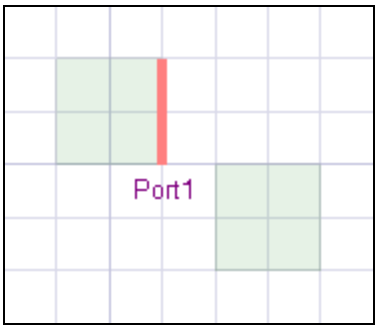
1. Right-click in the **Layout Editor** and select **Port** > **Create Terminal Ports**.
2. Click anywhere in the **Layout Editor** to create a terminal port. The port automatically seeks out a reference.



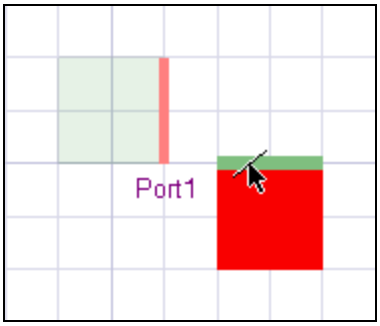
Assign Reference

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

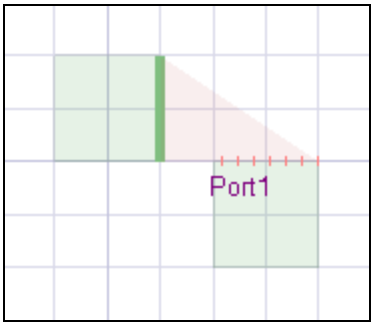
1. Select a port.



2. Enter edge selection mode and click the edge of an appropriate primitive.



3. Right-click in the **Layout Editor** and select **Port** > **Assign Reference** to use the primitive is used as a reference.



> **Note:**
>
> Refer to Assigning Reference Ports in the Layout Editor.

Clear Reference

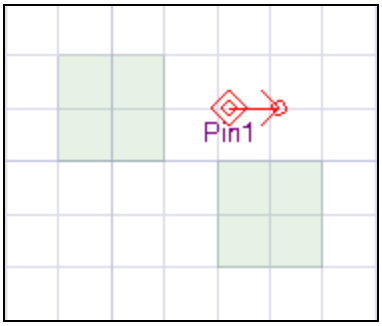
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select a port or visible reference.
2. Right-click in the **Layout Editor** and select **Port**> **Clear Reference**.

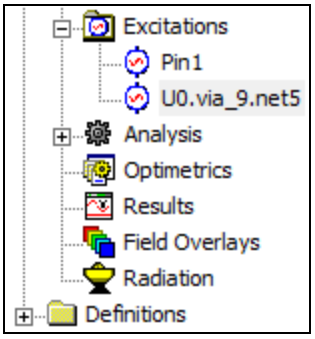
Refer to Assigning Reference Ports in the Layout Editor.

Create Ports on Net

1. Select a pin assigned to a net.



2. Right-click in the **Layout Editor** and select **Port**> **Create Ports on Net**. The new edge port is added to the **Excitation** folder in the **Project Manager** window.



> **Note:**
>
> Refer to Assigning Reference Ports in the Layout Editor.

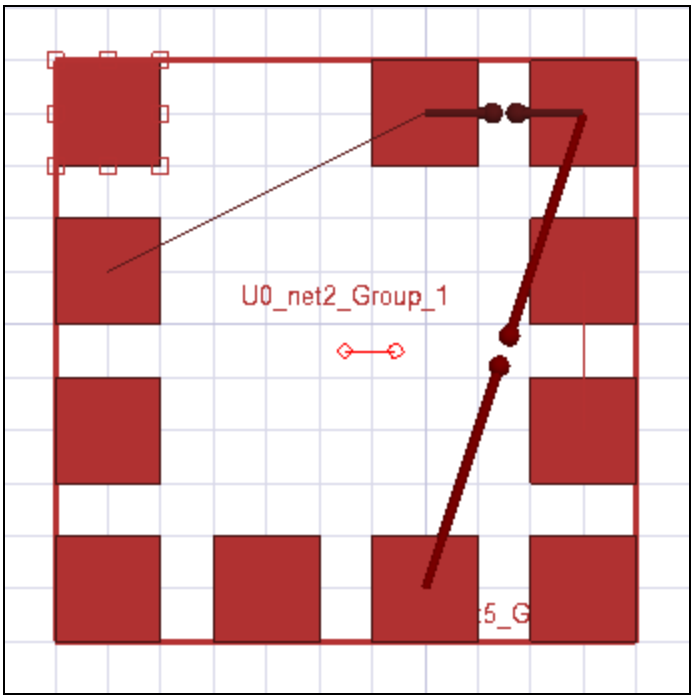
Remove Ports From Net

Right-click in the **Layout Editor** and select **Port** > **Remove Ports From Net**. The new edge port is deleted from the **Excitation** folder, along with the pin, and the pin is deleted from the **Layout Editor**.

Create Ports on Component

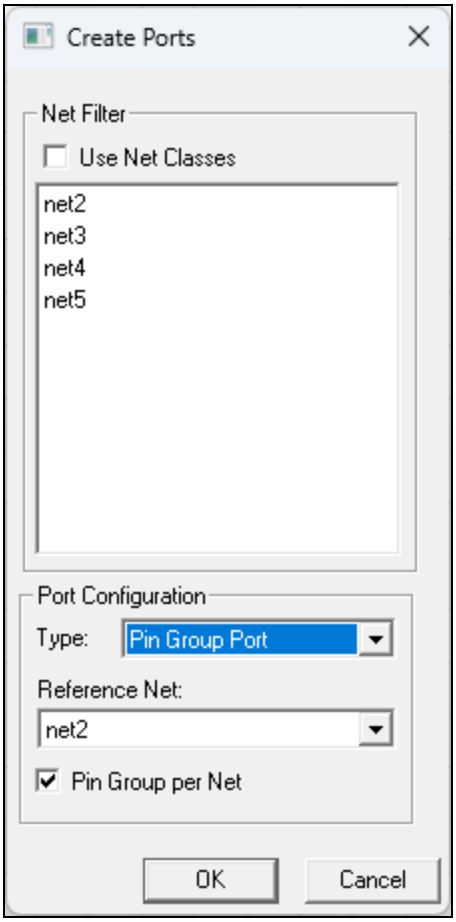
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Highlight a component.



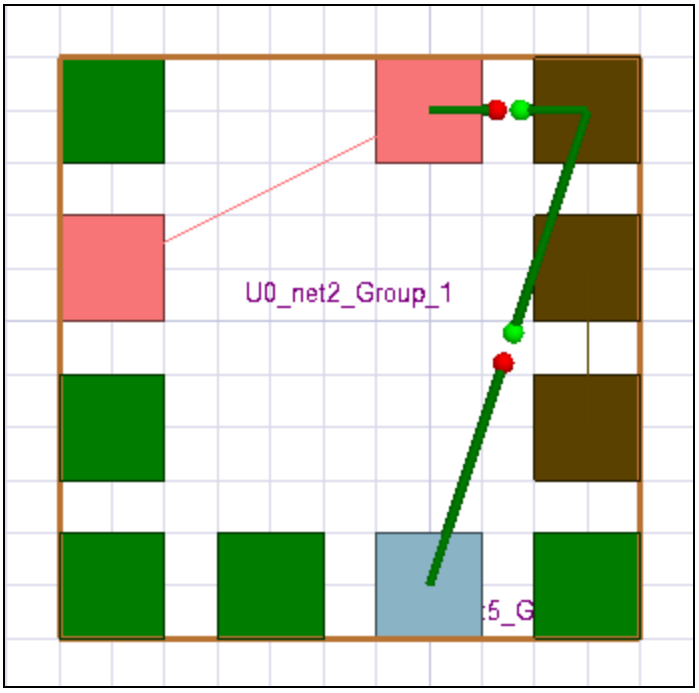
2. Right-click in the **Layout Editor** and select **Port** > **Create Ports on Component** to open the **Create Ports** window.

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 **Create Ports** window, do the following:
    - From the **Net Filter** area, check the **Use Net Classes** box, if applicable.
    - Select a net from the list.
    - From the **Port Configuration** area, select a **Type** of port from the drop-down menu (i.e., **Gap Coax Port**, **Circuit Port**, or **Pin Group Port**).
    - If **Pin Group Port** or **Circuit Port** is selected from the **Type** drop-down menu, the **Reference Net** drop-down menu appears. Select a **Reference Net** from the drop-down menu.
    - If **Pin Group Port** is selected from the **Type** drop-down menu, the **Pin Group per Net** check box appears. Check the **Pin Group per Net** box, if applicable.
4. Click **OK** and the new ports appear on 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.

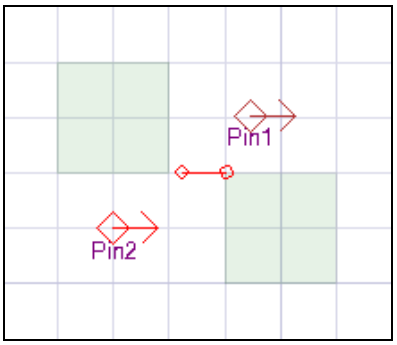


Remove Ports From Component

Right-click the appropriate component in the **Layout Editor** and select **Port** > **Remove Ports From Component**.

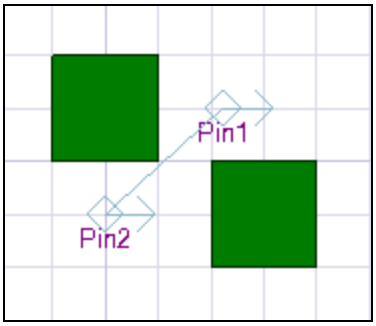
Create Pin Group

1. Select two or more pins.

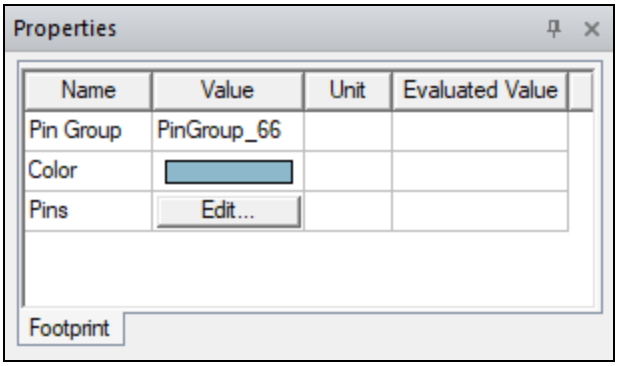


2. Right-click in the **Layout Editor** and select **Port** > **Create Pin Group** and a line is drawn between the pins on the **Layout Editor**, a visual representation of the new group.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



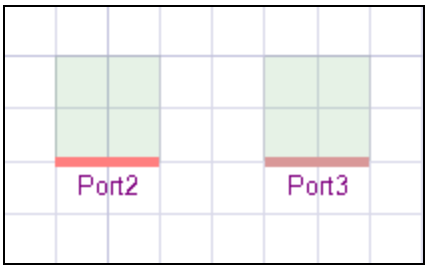
3. Select the line to view the group's details in the **Properties** window.



> **Note:**
>
> Refer to Creating a Pin Group.

Couple Edge Ports

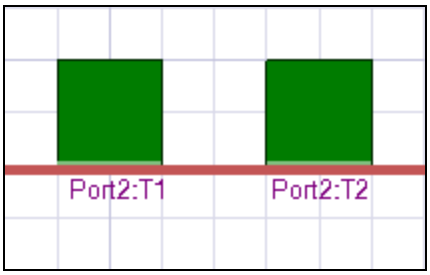
1. Select two parallel edge ports.



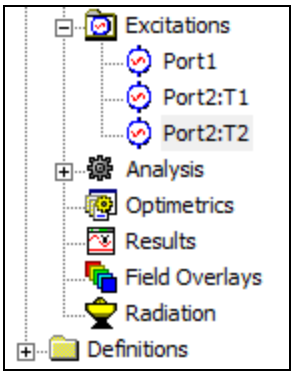
2. Right-click in the **Layout Editor** and select **Port** > **Couple Edge Ports** and a line is drawn under the pins on the **Layout Editor**, a visual representation of the new group.. The

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coupled ports retain the name given to the first selected port, which becomes Terminal 1. The second selected port becomes Terminal 2.



The coupled ports appear in the **Project Manager** window > **Excitation** folder with their newly coupled names (e.g., Port2:T1 and Port2:T2).



> **Note:**
>
> Refer to Creating Coupled Ports From Existing Edge Ports.

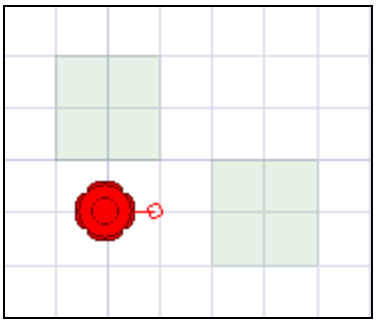
Decouple Edge Ports

Right-click the coupled edge port line in the **Layout Editor** and select **Port** > **Decouple Edge Ports**.

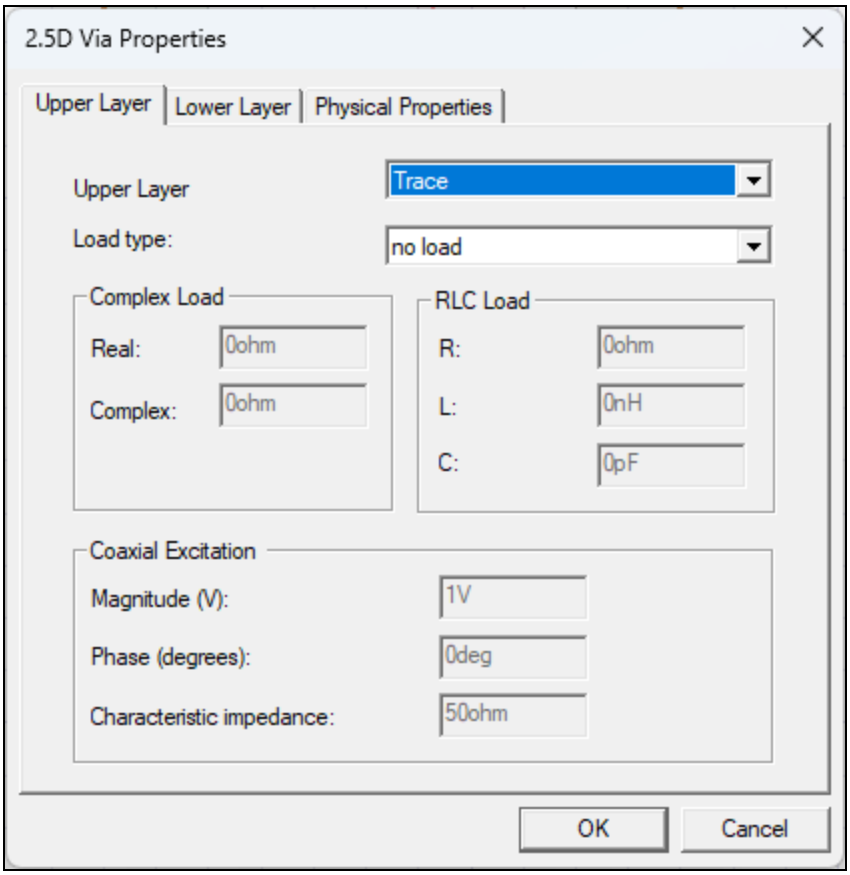
Coax Probe

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 via.

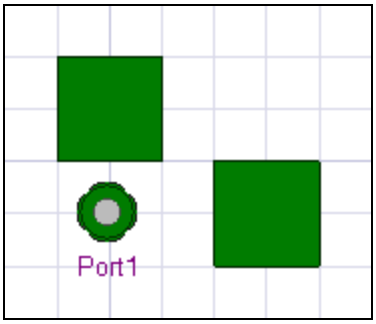


2. Right-click in the **Layout Editor** and select **Port** > **Coax Probe** to open the **2.5D Via Properties** window. Refer to Creating a Probe Port to configure the new probe port.



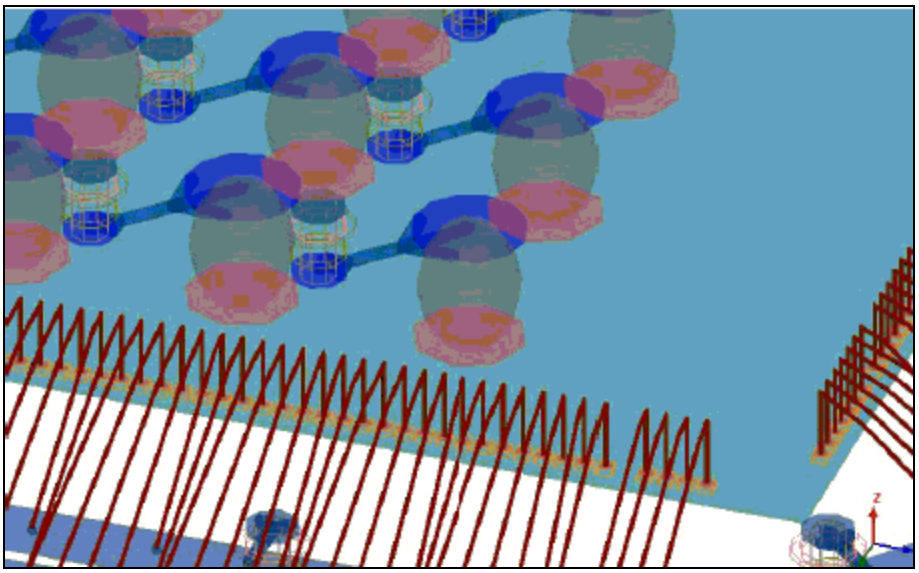
3. Click **OK** and the new probe port appears in the **Layout Editor**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Gap(coax) Ports

The Gap(coax) lumped port is commonly used to excite terminals at package interfaces. Common applications include excitation of vias, bondwires, and solderballs. Ports create circular geometry, the 2D cross-section for which should resemble a coaxial conductor.



The following pre-simulation validation checks apply:

- Port geometry does not intersect other port geometry
- Circumference is wholly contained by conductor

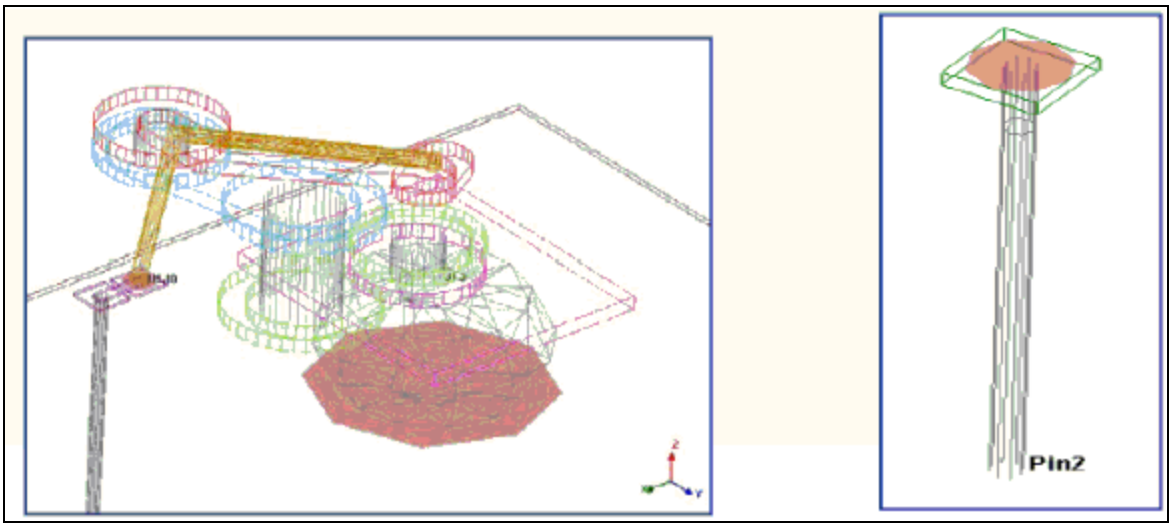
Note that, unlike other port types, terminal identification is performed post-mesh by the solver.

Terminal Assignments in Gap(coax) Ports

Because the terminal geometry of Gap(coax) ports often originates from unrelated and/or 3D structures, their identification and most validation checks are deferred to post-mesh. The port-

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solver searches for a valid terminal that is interior to the port-face in order to generate a selected field distribution. Various checks are included, and accurate messaging is provided. If applicable, pay particular attention to the **LayerAlignment** property, since the port face must contain the terminal geometry.



Wave(coax) Ports

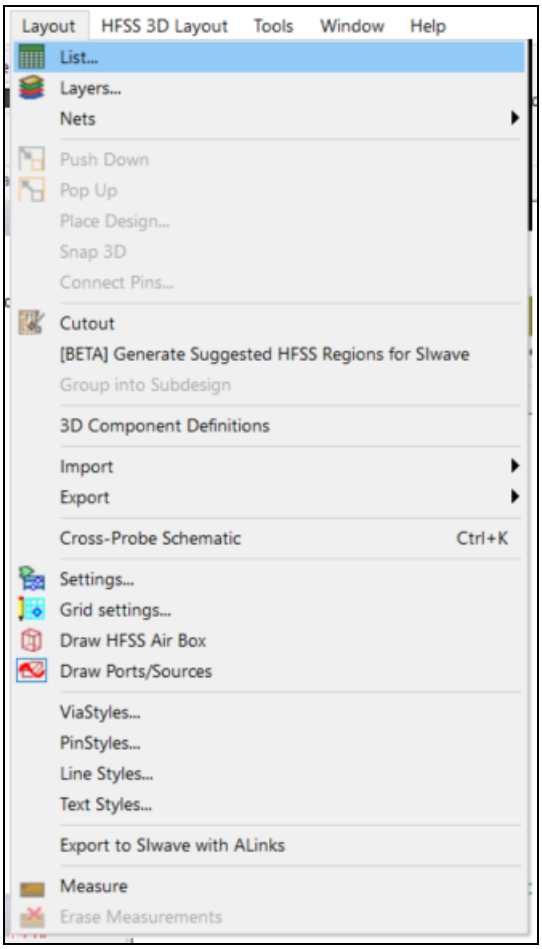
Coaxial wave ports can be used to excite solderballs at component interfaces. A wave port represents the region through which electromagnetic energy enters or exits the solution space. In the Ansys Electronics Desktop (AEDT)™ electronics systems design platform, a wave port is treated as if it were a semi-infinitely long waveguide or transmission line of the exact same cross-section attached to the model where it's excited. Wave ports assigned on waveguide structures are defined by the cross-section of the waveguides. For transmission lines (i.e. microstrip, coplanar waveguides, slotline etc.) ports should be defined carefully. Placement of a wave port is critical since it can affect the accuracy of the solutions.

If a coaxial wave port is placed on a component (e.g., a flip-chip), the reference plane is changed to a volume body with a thickness given by the "PEC Launch Width" property. This reference plane provides the required PEC backing for the wave port. Otherwise, the port is modified based on launch-width and becomes the Perfect Electric Conductor (i.e., PEC) instead.

Follow these steps to create a wave port using an existing Gap(coax) port.

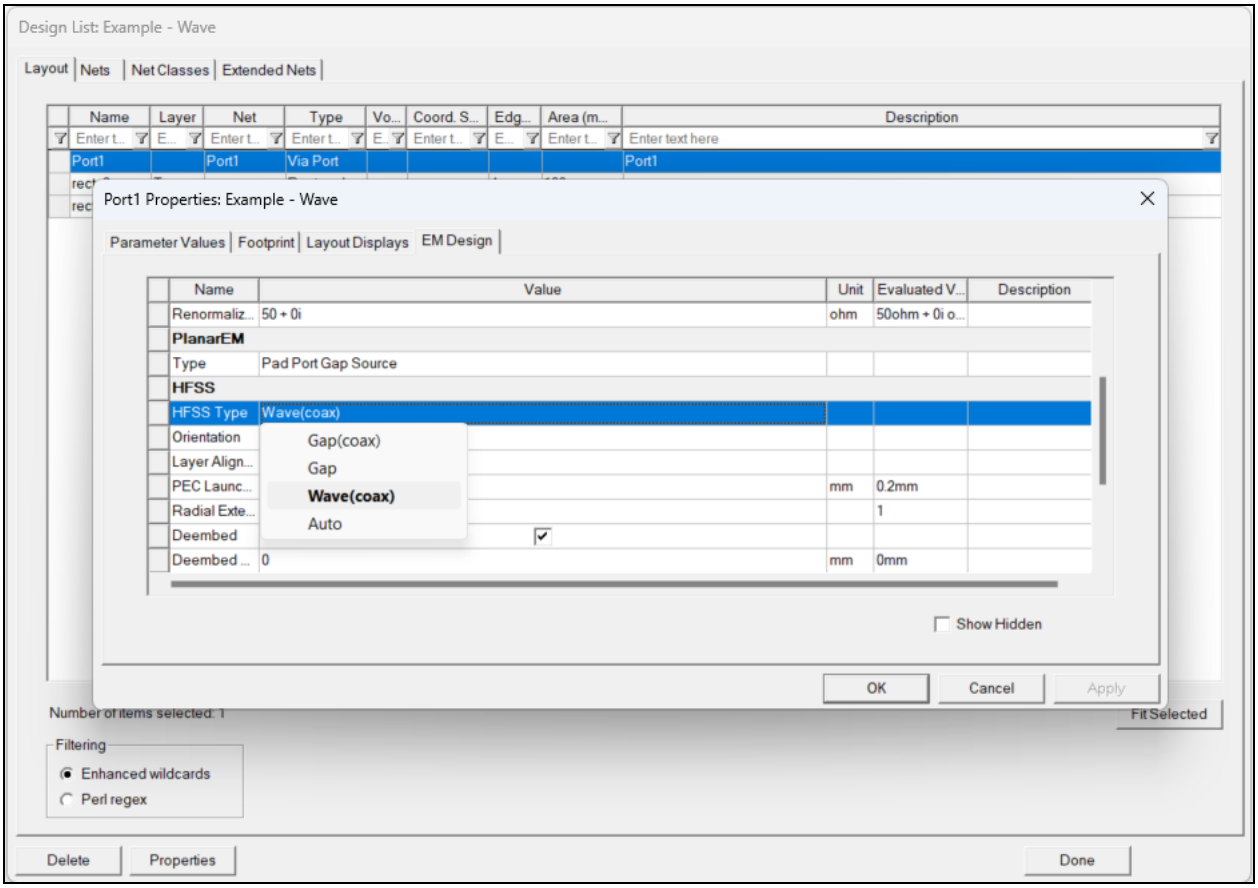
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Navigate to **Layout** > **List** to open the **Design List** window.



2. Select a port from the table and click **Properties** to open its individual **Properties** window.
3. From the **HFSS** section of the **Parameter Values** tab, select **Wave(coax)** from the **HFSS Type** 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. Click **OK** to close the **Properties** window.

Horizontal Wave Ports

Horizontal wave ports provide the same excitation mechanisms as vertical wave ports for package and PCB interfaces. These ports are typically created on voids or negative space, where the perimeter becomes the reference. Interior objects within the void become wave port terminals. While most examples use one or two terminals, any number of terminals can be defined.

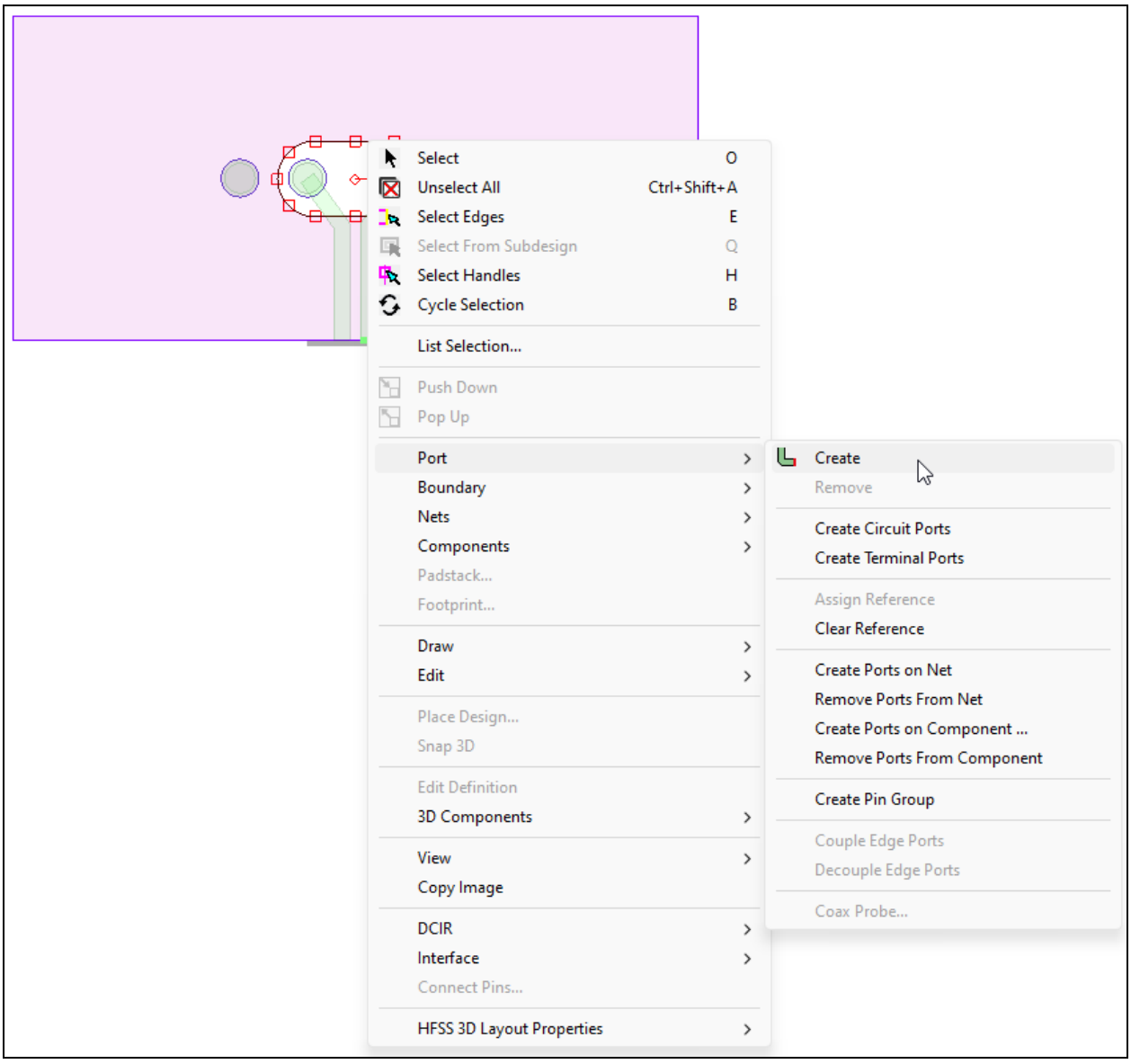
Follow these steps to create and configure a horizontal wave port:

1. From the **Layout Editor**, right-click within negative space (i.e., a void/antipad) and select **Port** > **Create**. The new port appears in the **Project Manager** window within **Project** > **active design** > **Excitations** as two terminals (e.g., `Port2:neg` and `Port2:pos`).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

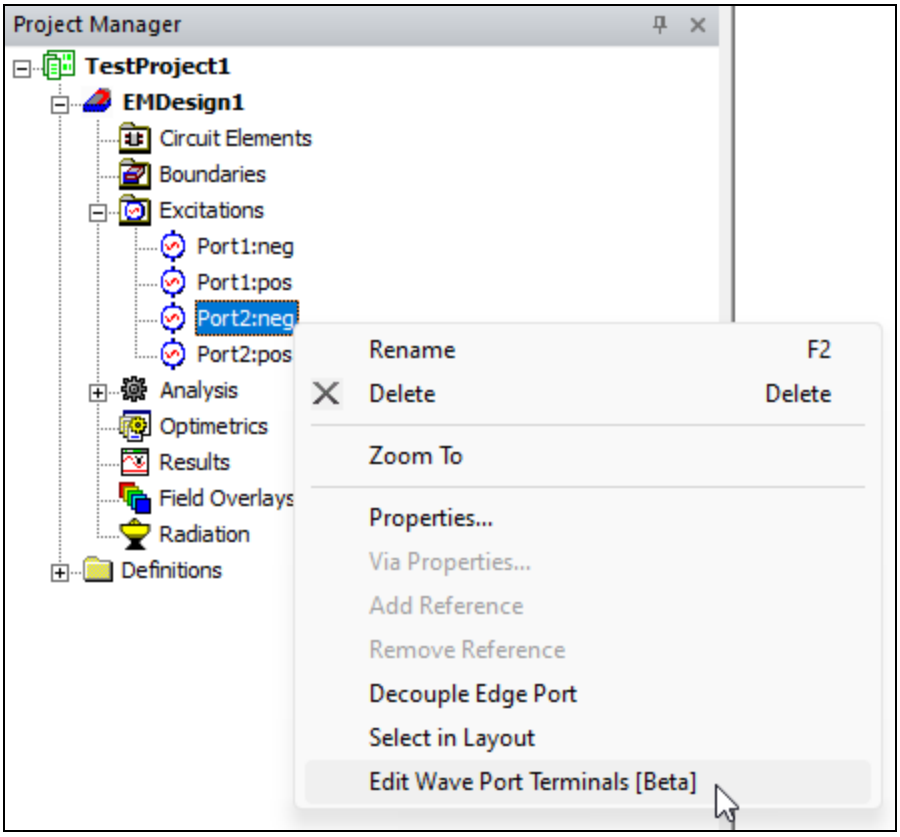
Note:

Any interior objects inside the void are automatically assigned as terminals.

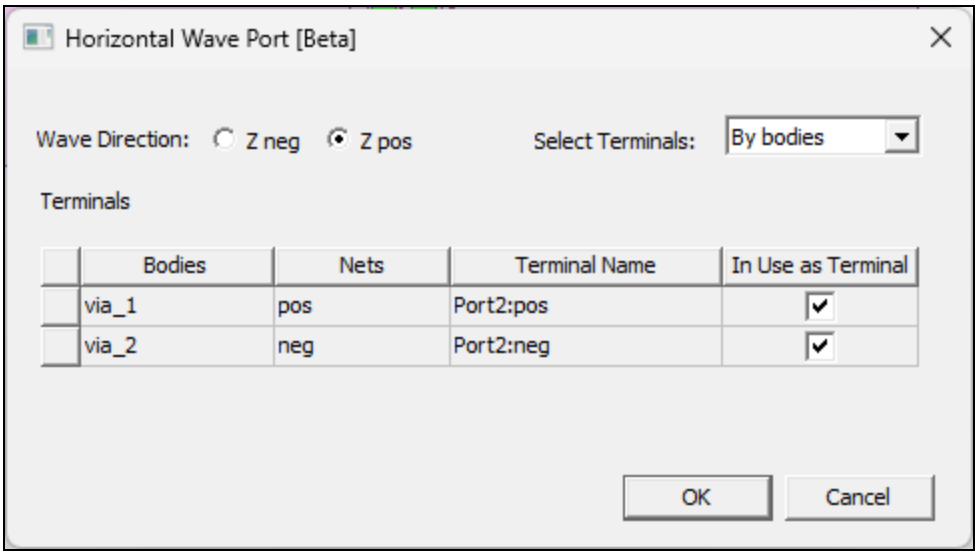


2. Right-click either terminal and select **Edit Wave Port Terminals** to open the **Horizontal Wave Port** window.

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 **Horizontal Wave Port** window, configure 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.

- **Wave Direction** – select **Z neg** or **Z pos** to update the layer alignment of all terminals in the same horizontal wave port definition.
- **Select Terminals** – from the drop-down menu, choose **By bodies** or **By nets** to reorganize the table by either category.
    - **By bodies** – reorganize terminals alphabetically by body names, most common when the port is created from voids or antipads.
    - **By nets** – reorganize terminals by net names, for connectivity-based assignments where terminals are defined by net associations. If no net information is available, the Nets column will display *NO-NET*.
- In the **Terminals** table, uncheck a box in the **In Use as Terminal** column to deactivate the adjacent terminal. Although the geometry within the port is no longer considered a terminal, the geometry is not removed from the port. Floating geometry can impact results.

> **Note:**
>
> If a terminal is deactivated, the geometry within the port is no longer considered a terminal. However, the geometry is not removed from the port. Floating geometry can impact results.
>
> Adding new terminals cannot be done through the **Terminals** table. To add terminals or modify the orientation of terminals, recreate the port.

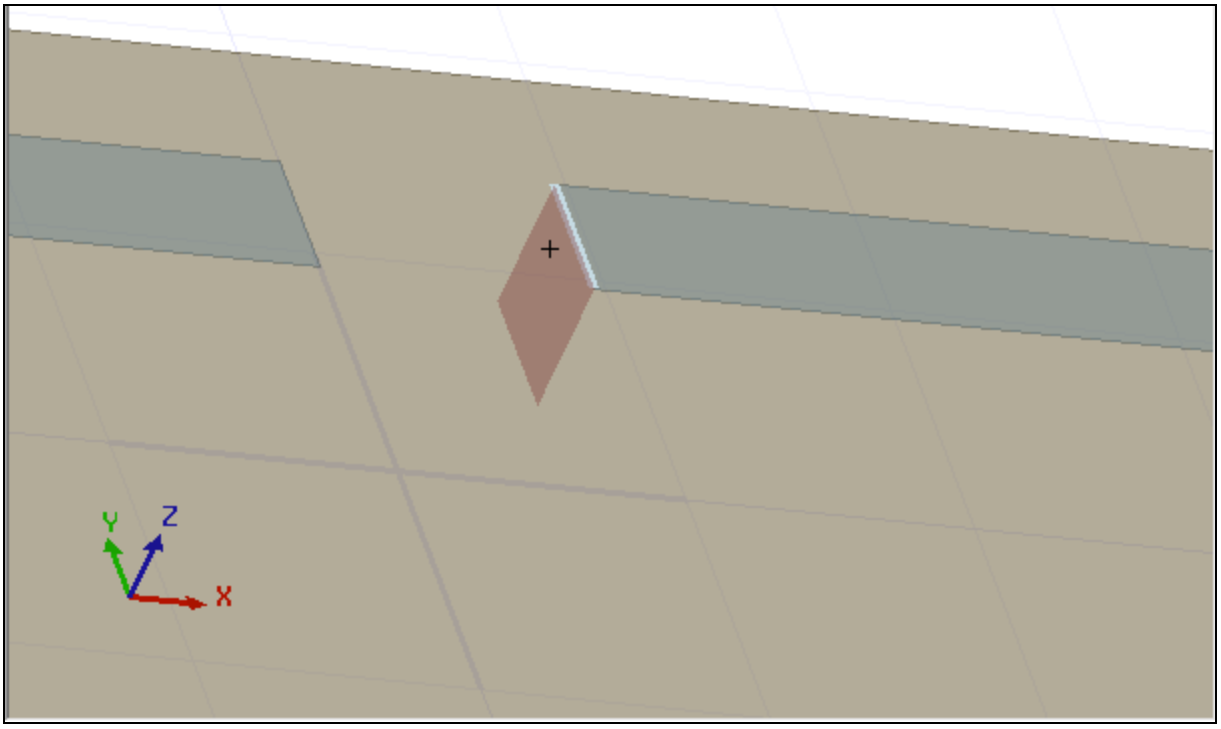
Horizontal Lumped Ports

It is possible to configure an HFSS port to be a horizontal lumped port. Any existing horizontal edge port or pin may be assigned a reference segment. Reference assignment for an edge port requires a coplanar and parallel edge to realize valid port-geometry, but reference assignment to a pin can tolerate some geometry overlap.

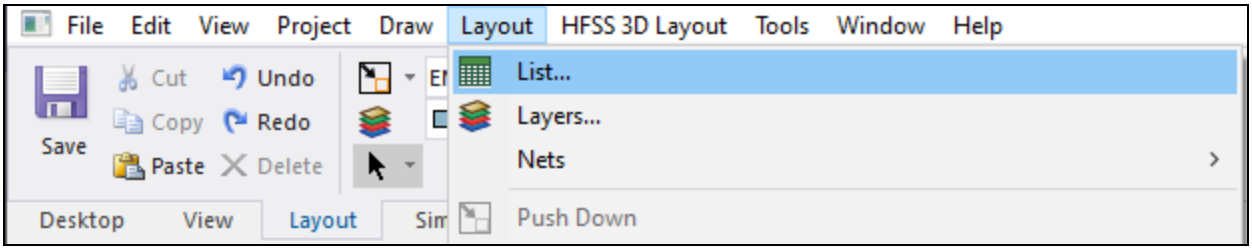
Follow these steps to create a horizontal lumped port.

1. Do one of the following to select the vertical edge port you want to assign a reference segment:
- From the **Layout Editor**, click the port with the **Select** cursor (i.e., the default cursor, which resembles cross hairs in the **Layout Editor**).

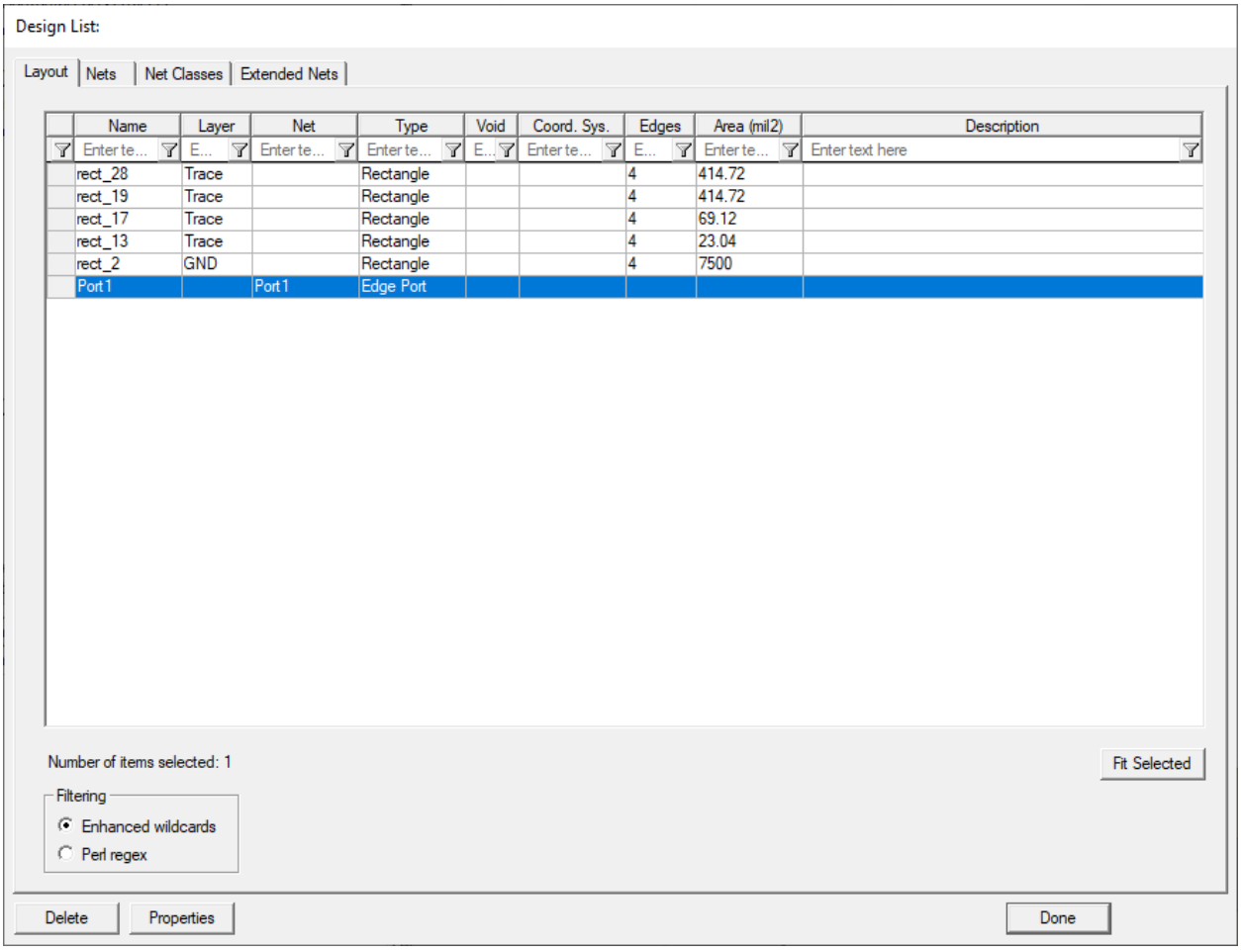
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



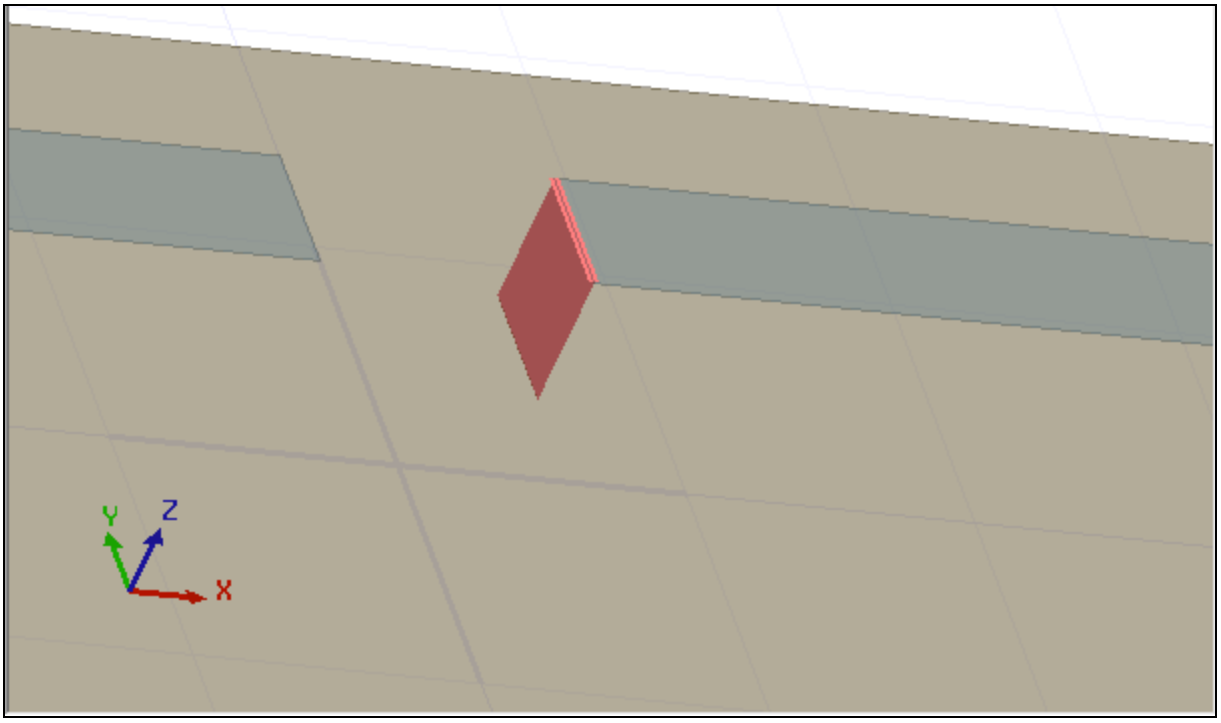
- From **Layout**, select **List** to open the **List** window. Then click anywhere within the port's row to highlight it. Click **Done** to close the **List** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



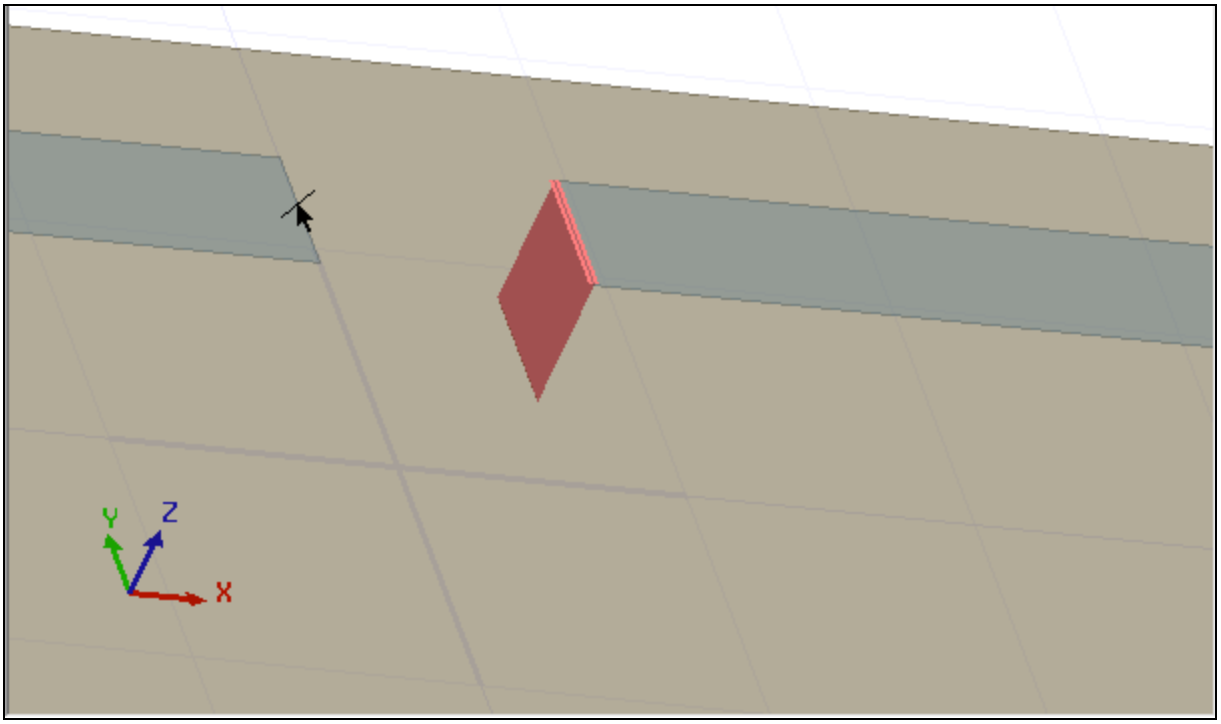
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



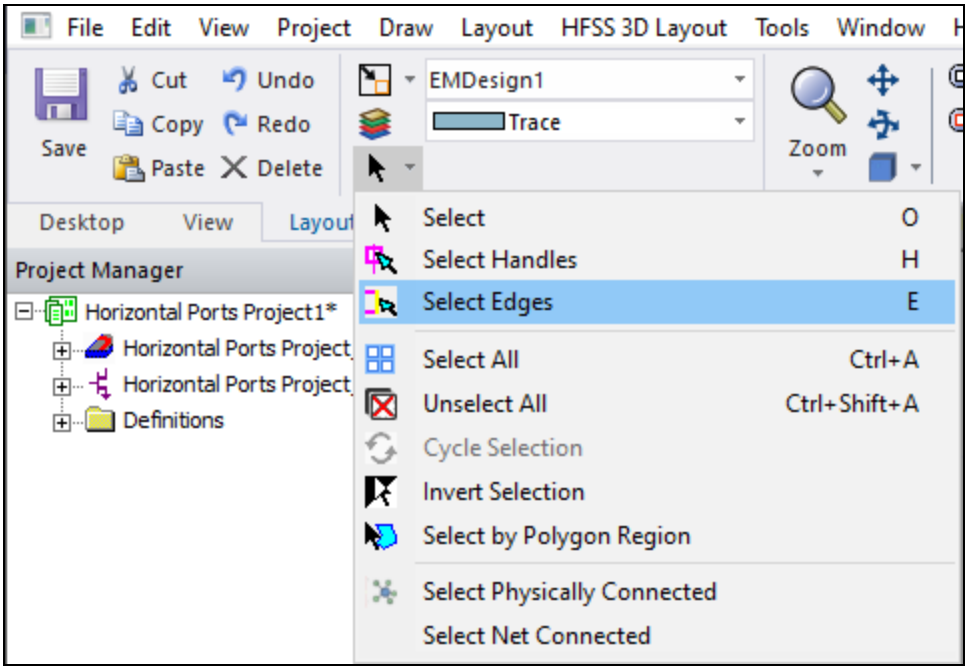
2. Do one of the following to select the reference edge:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

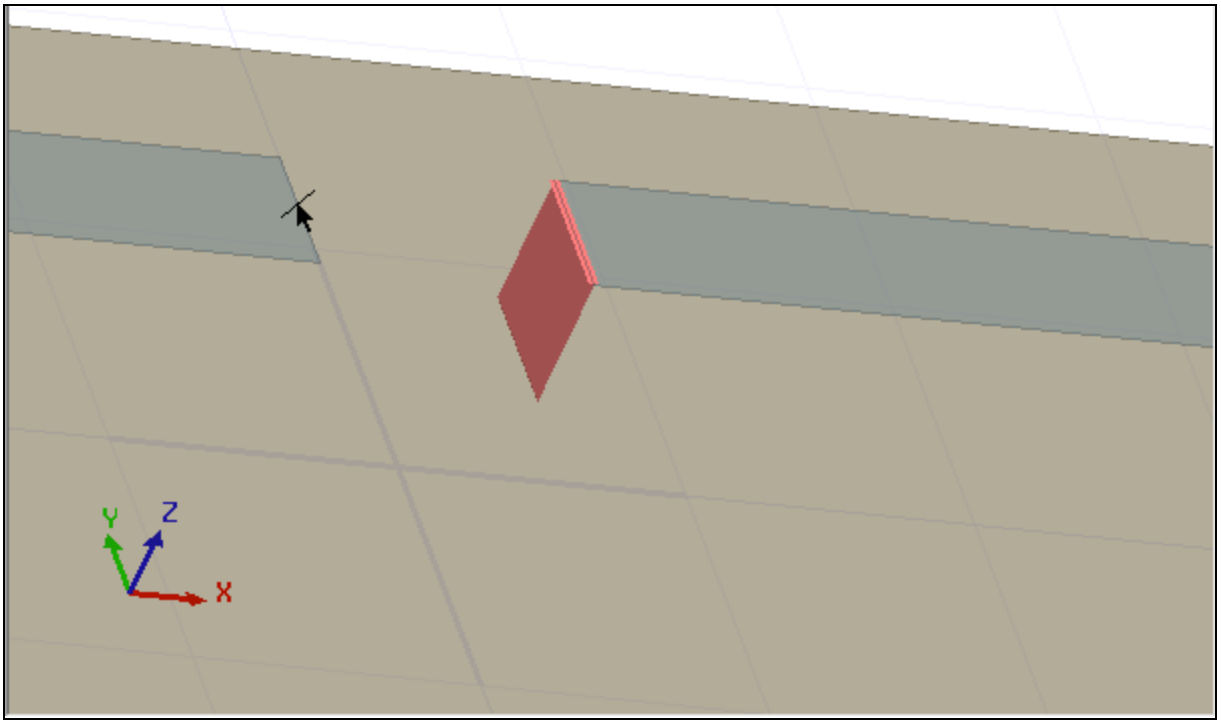
- Press **E** to change the cursor to the edge selection cursor. Then click the reference edge.



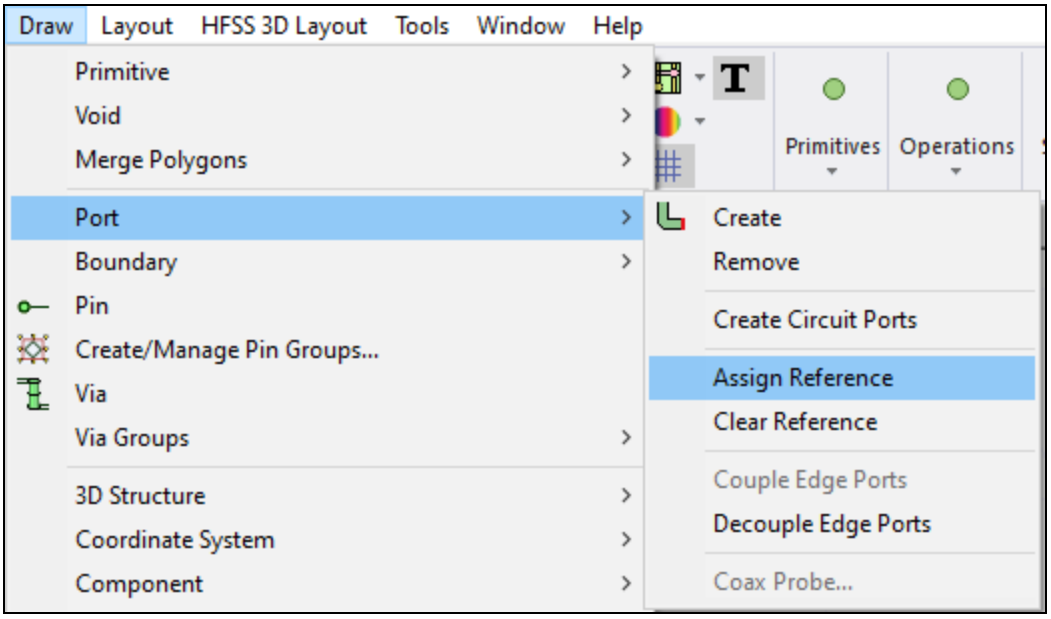
- From the **Layout** tab, choose **Select Edges** on the cursor drop-down menu. Then click the reference edge.



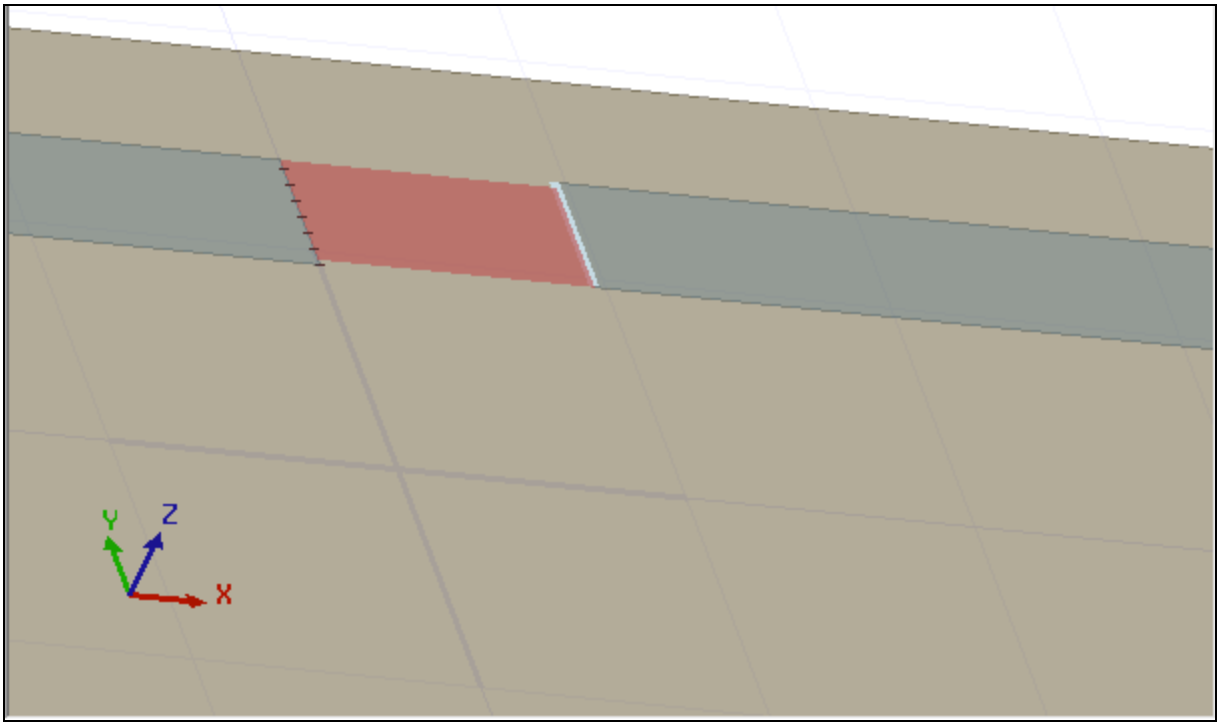
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. From **Draw**, select **Port** > **Assign Reference** to create a horizontal lumped port.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Deembedded lumped ports in HFSS 3D Layout are not restricted to being rectangular. They can be arbitrary in shape. A typical non-rectangular lumped port are trapezoidal, with parallel sides connecting to the terminal and its reference. But more complicated shapes are allowed.

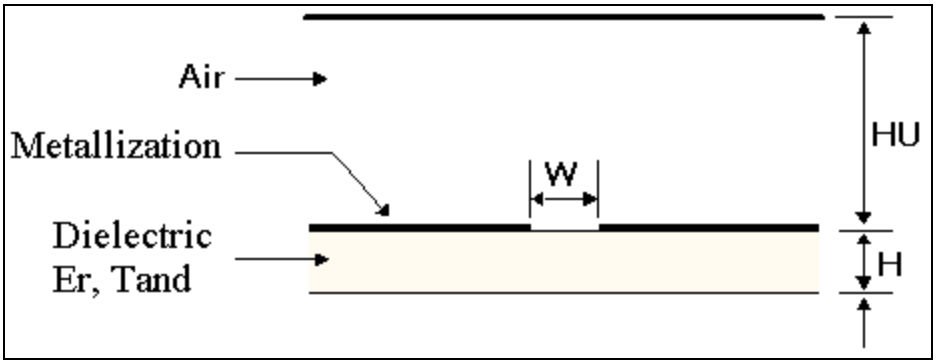
To tune the solver to reliably solve low frequencies for designs that only contain lumped ports as sources, see the **Enhanced low frequency accuracy** setting in the Options Tab of the **HFSS Setup** window.

CPW and Slotline Ports

CPW and Slotline Ports are allowed and can be configured in light of the following information.

- On a Negative Signal layer, edge selections must be co-linear and co-planar.
- The CPW mode can also be solved on a Signal layer by selecting only the single conductor and sizing the port appropriately.
- Various configurations using lumped-gap ports are possible.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Circuit Ports and Elements in HFSS 3D Layout

Significant simulation speed and efficiency can be gained by using circuit ports and elements in HFSS. Reduce simulation times by limiting the number of excitations by using circuit elements to include black-box representations directly in HFSS solutions.

Circuit Element Advantages

A circuit port consists of an impressed current source along with a user-defined parallel impedance load. Compared with traditional ports in HFSS 3D Layout, circuit ports are easier to setup but still provide S-parameters.

- The only geometry that must be specified is the location of two terminals.
- A terminal can be defined as a selected edge or as a point touching a metallic object (e.g., an edge port or a point port).
- A design that only contains circuit ports can also be solved at much lower frequencies, compared with using standard ports.

Circuit elements provide a simple and flexible approach to include the effects of lumped circuit elements and allow you to include black box representations as S-parameters directly in an HFSS solution. One benefit of this approach is to reduce the number of excitations required. Reducing the number of excitations reduces the simulation time.

Deembedded Lumped Ports in HFSS 3D Layout are not restricted to being rectangular, they can be arbitrary in shape. A typical non-rectangular lumped port are trapezoidal, with parallel sides connecting to the terminal and its reference. But more complicated shapes are allowed.

A circuit element connection is modeled as if there is actual geometry carrying a conduction current but without the explicit need to define the geometry. This is important to keep in mind since there is an actual connection between two terminals being enforced at the discretized FEM system matrix, and that connection is being solved. Which is similar to the solution if there existed explicit geometry to model that connection. Similar to circuit ports, the only geometry that

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

must be specified is the locations of the terminals defining the circuit connections where each terminal is defined as a selected edge of a metallic object.

Restrictions and Workarounds

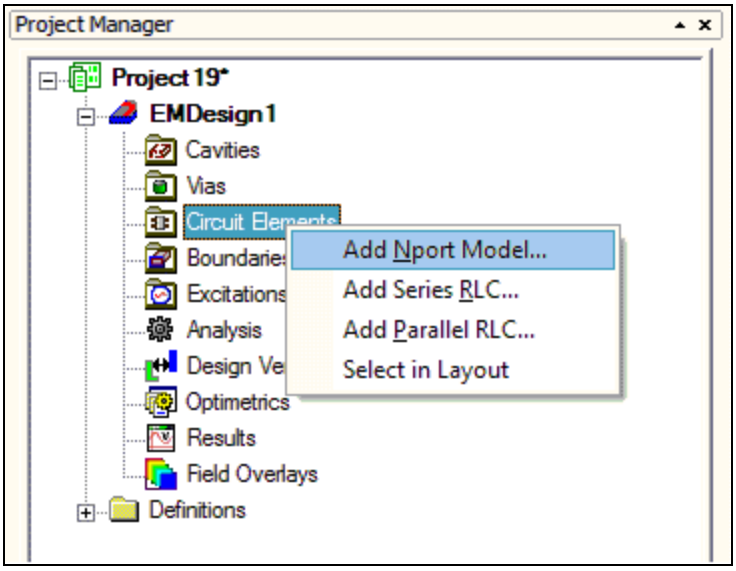
The only restriction on the placement of circuit elements and ports is there may not be any gap or wave ports located directly between the terminals of the circuit element/port. While adding a circuit element/port to a design does not explicitly add any conductive material between the terminals, it does add a current path which interferes with any gap or wave ports in between. Such intersections are usually an unintended consequence of using a highly conductive circuit element to add a current return path to a design and can usually be avoided by one of four methods:

- The intersecting wave ports or gap ports can be switched to circuit ports.
- The circuit element can be re-routed by moving the terminals.
- Two separate circuit elements can be used to connect the terminals to a common ground plane or an explicit return path in a separate signal layer.
- If the intersecting ports are horizontal gap ports in a signal layer with non-zero thickness, they can be switched to align with the upper surface of the layer.

HFSS SoD

HFSS SoD supports simulation of two-port models, as well as Parallel and Series RLC ideal components. Additionally, HFSS 3D Layout can simulate N-port models. The HFSS solver models two terminal circuit elements by assuming a constant impressed current between the two terminals. HFSS 3D Layout also assumes this current has no radiative coupling to the structure. These assumptions are valid when the separation between the two terminals is electrically short, which is typically the case for such devices as surface mount resistors and capacitors. To add one of these components, right-click 'Circuit Elements' and select 'Add Nport Model', 'Add Series RLC' or 'Add Parallel RLC'.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

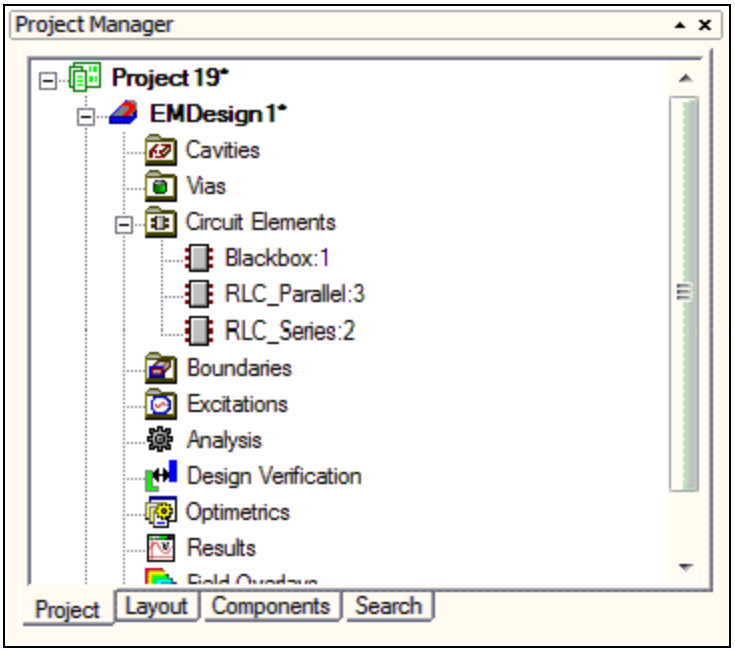


Select **Import Component Mapping File** or **Import Component File**.

- **Import Component Mapping File** opens a window to select a *.dat partmap file. HFSS reads the partmap file and create (or re-assign) and orient circuit-elements for given components.
- **Import Component File** opens a window to select a *.cmp file. HFSS reads this cmp file and create (or re-assign) circuit-elements.

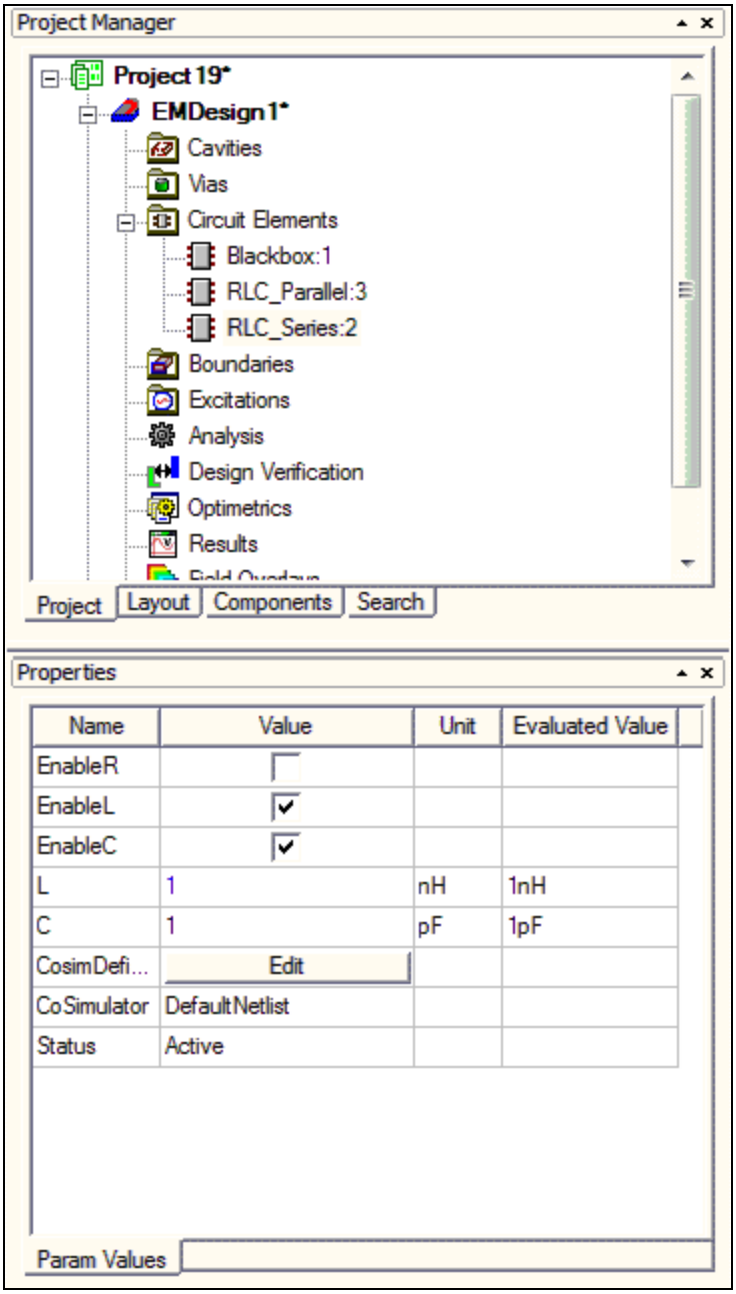
Once circuit elements have been added to an HFSS 3D Layout they appear under the Circuit Elements tree item.

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 the RLC components can be edited in the properties window and R, L and C can be deactivated.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



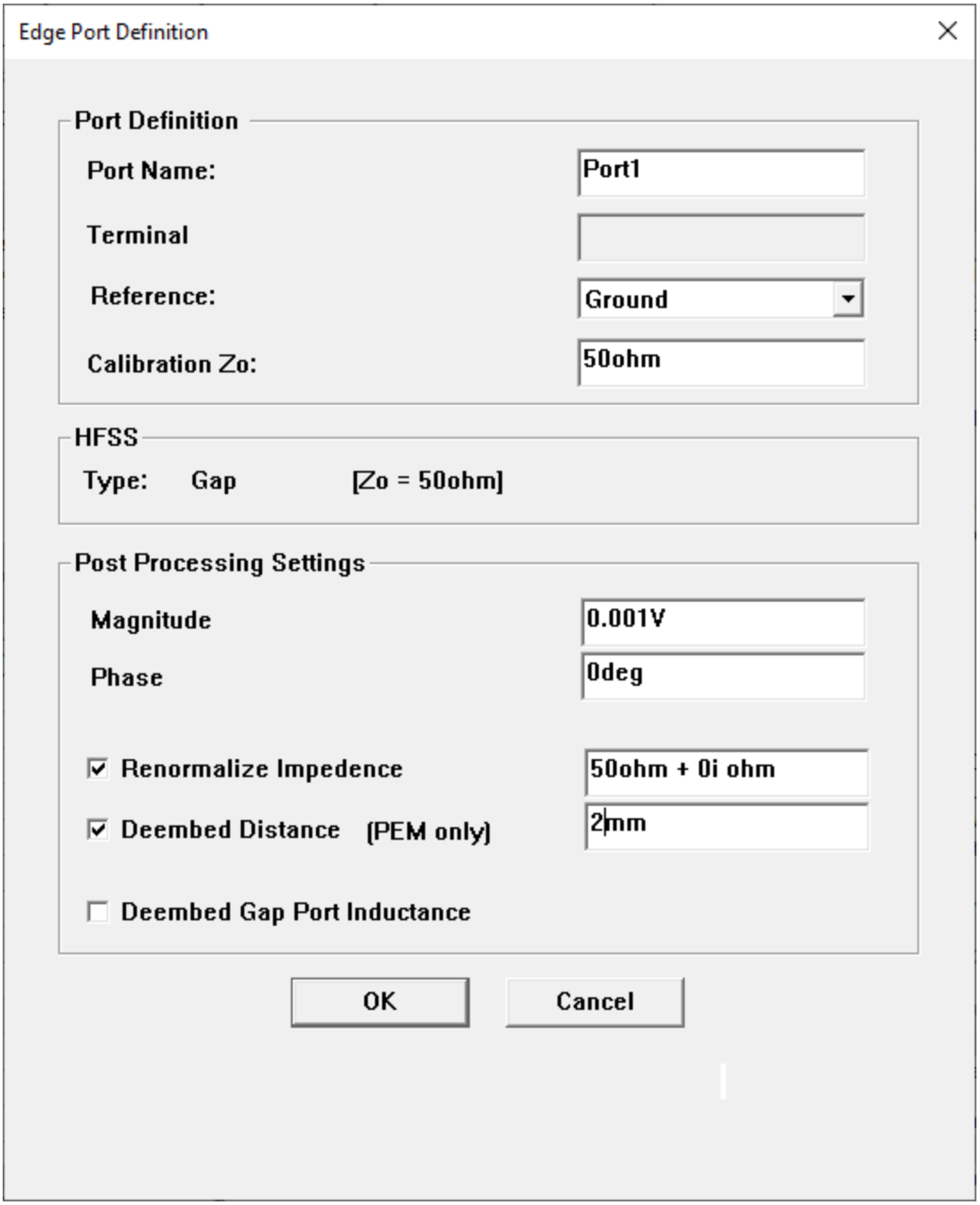
Note:

Components that do not have footprint geometry are initially rendered on the Symbol layer. When the component is first wired to an object on a physical layer, it renders on that physical layer. For more information see Components with a Symbol Footprint.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

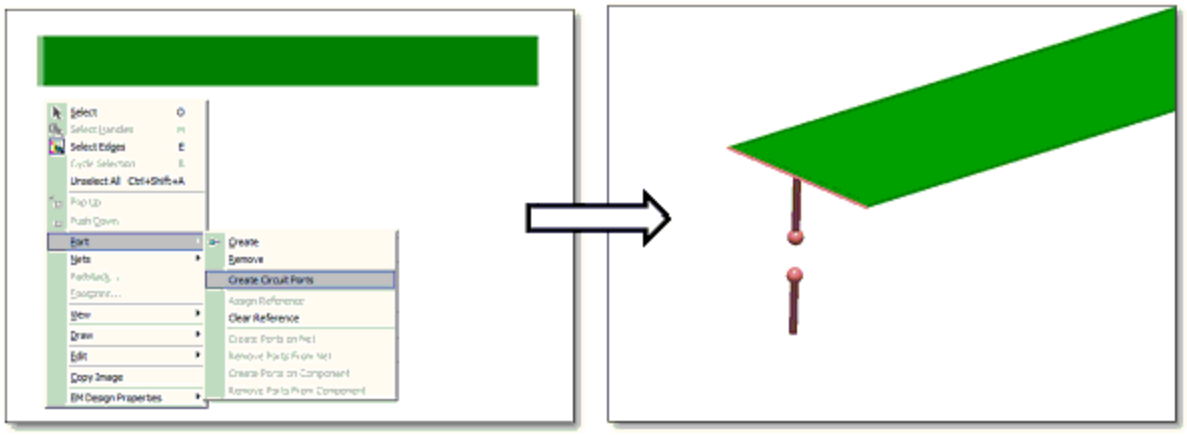
Creating a Circuit Port

Circuit ports can be created in multiple ways: automatically referenced to the closest ground plane, between two edges, or point-to-point. All circuit ports are listed under **Excitations** in the **Project Tree**. To edit a circuit port's properties, double-click the port in the **Project Tree** to open the **Edge Port Definition** window.



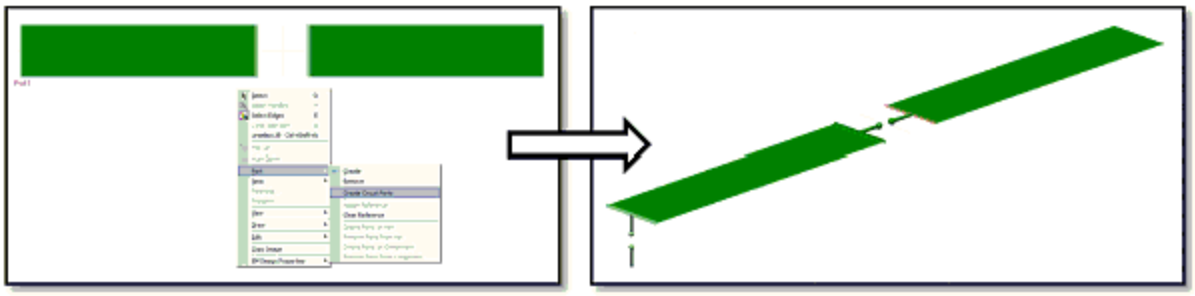
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create a circuit port automatically referenced to the closest ground plane, select an edge of a conductor, right-click and select **Port** > **Create Circuit Ports** on the floating menu.



A circuit port are created with the selected edge as its positive terminal and the nearest ground plane as its negative terminal.

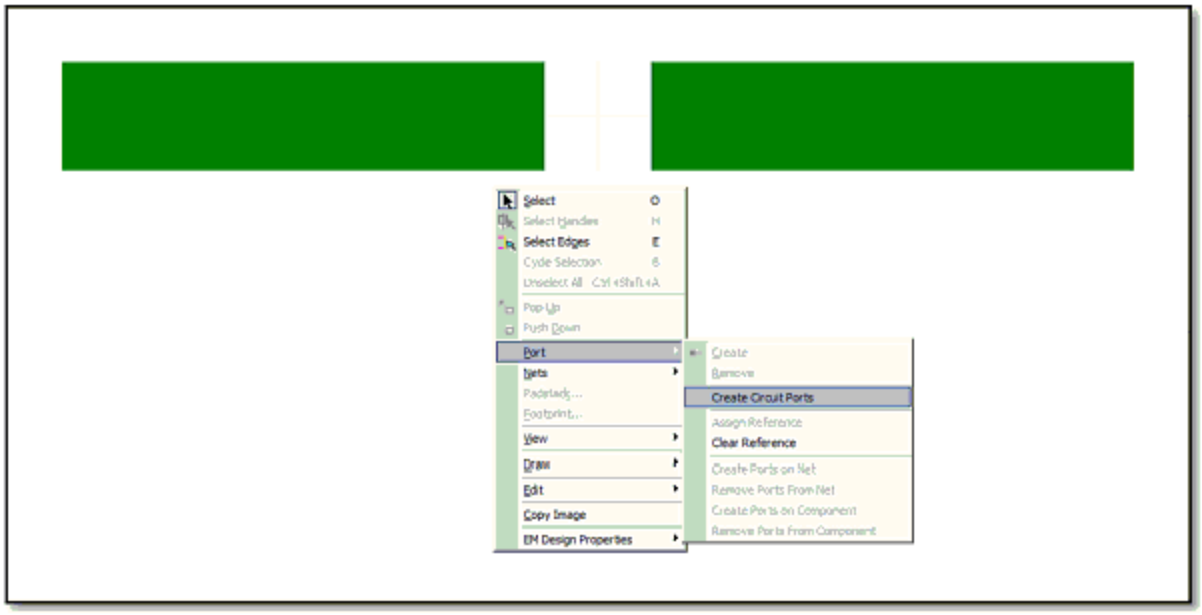
To create a circuit port between two edges, select a pair of edges from different conductors, right-click and select **Port** > **Create Circuit Ports** on the floating menu.



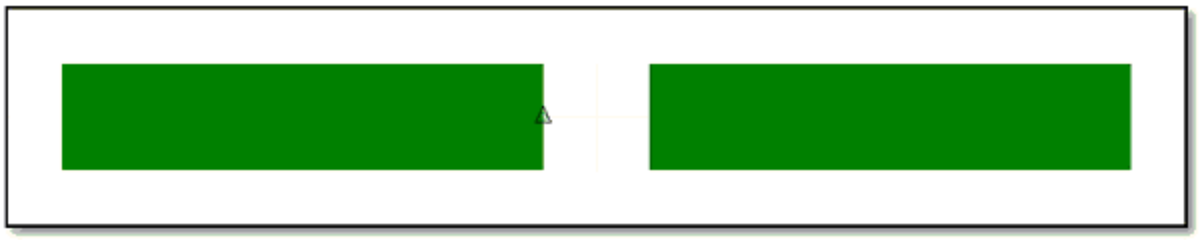
The first selected edge defines the positive terminal and the second selected edge defines the negative terminal of the port.

Circuit ports can also be introduced in a point-to-point manner. With no edges selected, right-click and select **Port** > **Create Circuit Ports** on the floating menu.

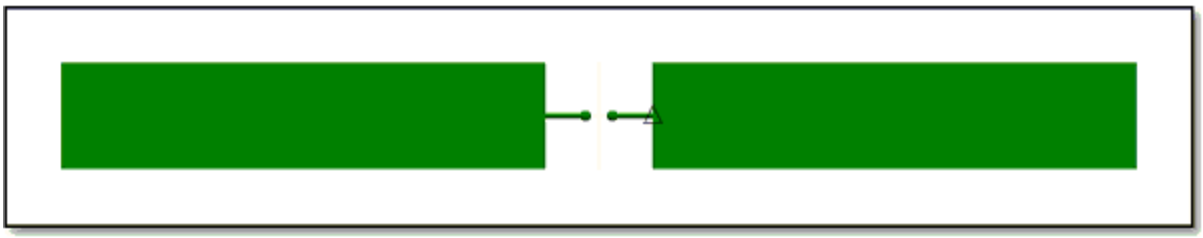
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click on a conductor to select the positive terminal.

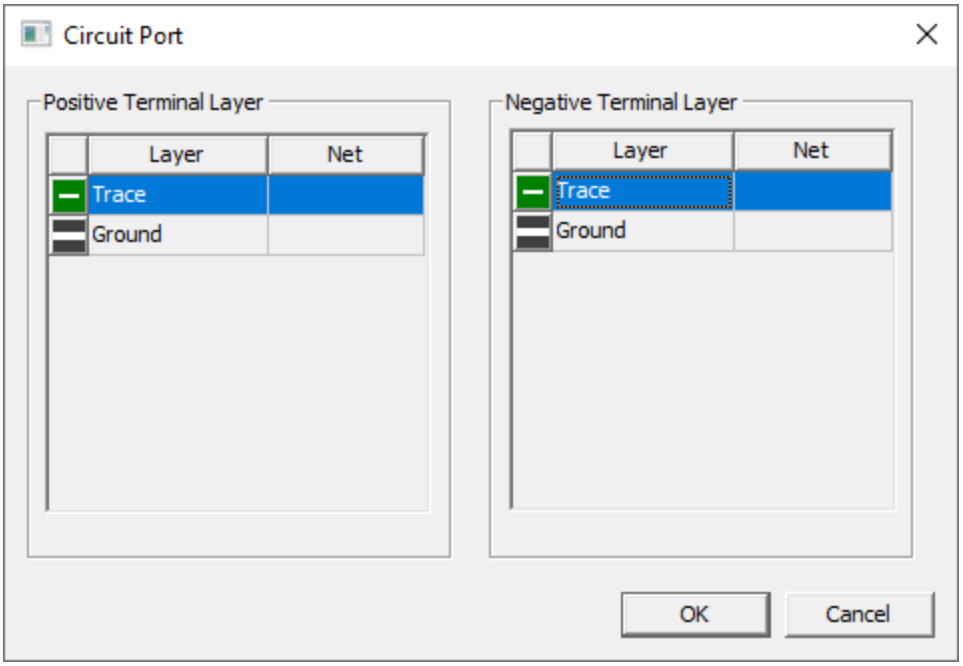


Click on a second conductor to select the negative terminal.



A window appears that allows the selection of a layer for each terminal.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The circuit port is listed under **Excitations** in the **Project Tree**. To edit the port's properties, double-click the edge port in the **Project Tree**. The **Edge Port Definition** window opens where you can perform these tasks:

- Change the name of the pad port.
- Deembedding S-Matrices.
- Vary the current excitation.
- Renormalize the port to a specific port impedance.
- Setup a Reference to the Nearest Negative Signal.

When the circuit port is selected, its **EM Design** tab is available in the **Properties** window. This tab displays many of the values in the **Edge Port Definition** window, including the port classification for the HFSS-PI type. The **Footprint** tab lists whether the circuit port is a point or edge port.

A port can be split across multiple signal layers. It can be made of up to three different line segments on three different primitive objects. Ports located over apertures in negative signal planes can generate incorrect results. HFSS automatically displays a warning message if referencing a gap source to the nearest negative signal produces such an error.

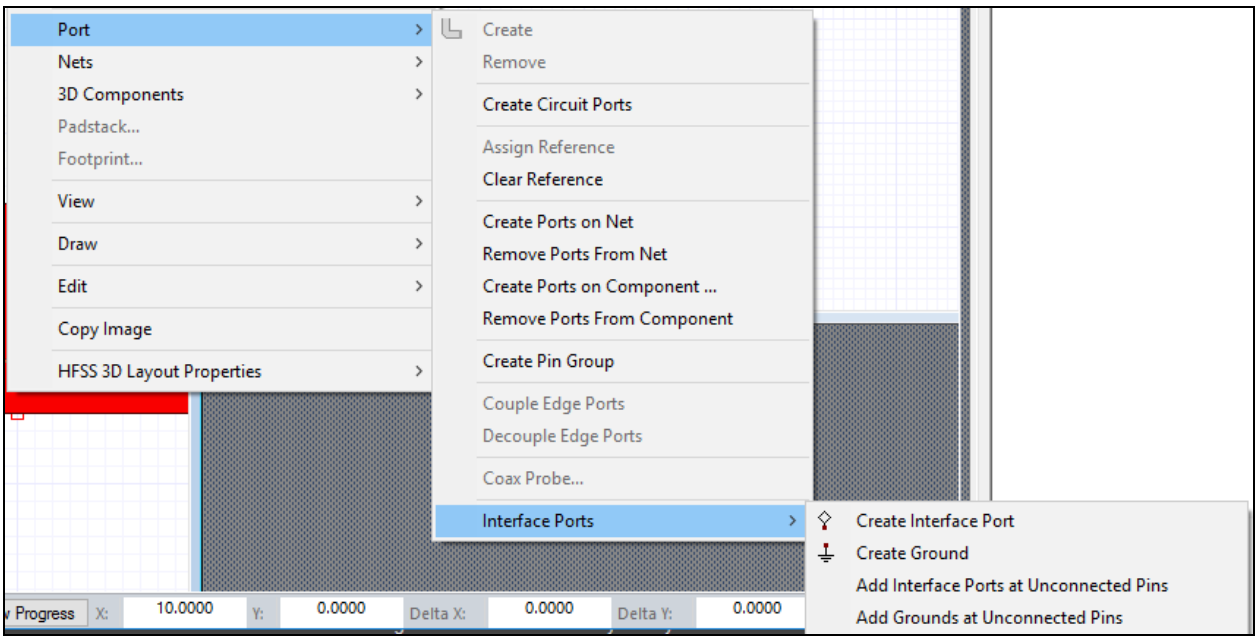
Interface Ports and Sources in 3D Layout

Interface ports and sources (interface and ground) can be added to a 3D Layout design. This applies to any Nexxim solution in 3D Layout (Transient or LNA). Creating interface ports and

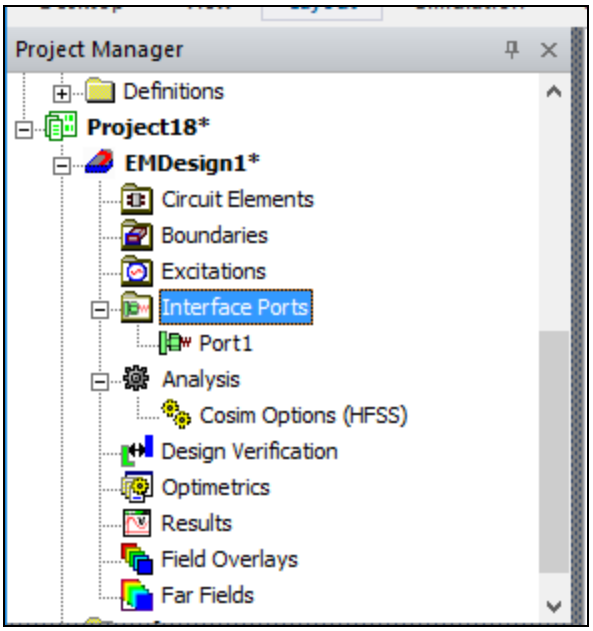
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sources is appropriate to drive Nexxim simulation from 3D Layout.

To place an interface port, click the toolbar icon or right-click in the **Layout Editor** and select **Port** > **Interface Ports** > **Create Interface Port**.

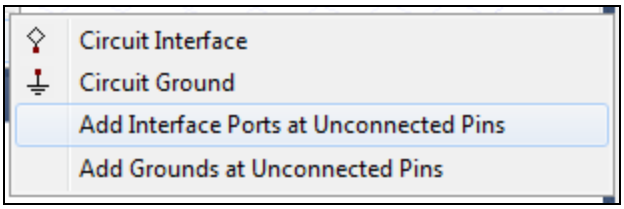


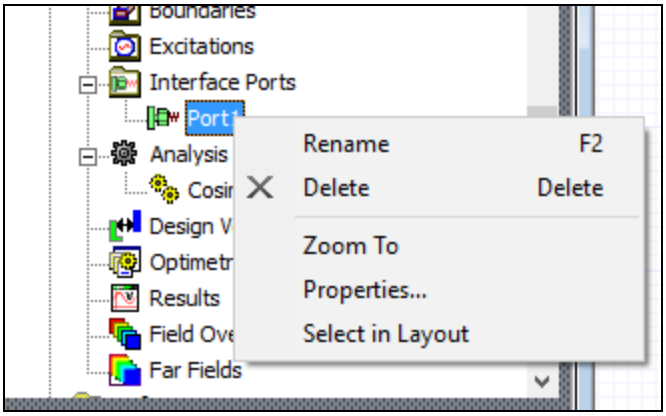
Interface and ground ports can be connected to 3D Layout ports.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Add Interface Ports at Unconnected Pins**: Automatically create interface ports on unconnected port instances for the selected component(s).
- **Add Grounds at Unconnected Pins**: Automatically create circuit grounds on unconnected port instances for the selected component(s).





Interface Ports

Once interface ports are placed in layout, an Interface Ports item appears in the **Project Tree**.

- **Properties**: Invokes the Interface Port window; users can add interface sources from this window.

Interface Sources

When interface sources have been added to ports, they appear under this **Project Tree** item.

Netlisting for Interface Ports and Grounds

To see how ports and grounds are netlists, RMC on the LNA or Transient setup to browse the netlist. Netlisting for ports, sources, and grounds is similar to that of a 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.

```
20   RPCB_Port1 Port1 0 PORTNUM=3 RZ=50 IZ=0
21   .PORT Port1 0 3 RPCB_Port1
22   RPCB_Port2 Port2 0 PORTNUM=4 RZ=50 IZ=0
23   .PORT Port2 0 4 RPCB_Port2
24   RU1_via_3_n1 U1_via_3_n1 0 PORTNUM=1 RZ=50 IZ=0
25   .PORT U1_via_3_n1 0 1 RU1_via_3_n1
26   RU1_via_4_n2 U1_via_4_n2 0 PORTNUM=2 RZ=50 IZ=0
27   .PORT U1_via_4_n2 0 2 RU1_via_4_n2
28
29   * end toplevel circuit
30
31   .TRAN 1e-010 1e-008
32
33
34   .end
```

Changing the Reference Surface

HFSS 3D Layout edge ports are usually single-terminal excitations, in which the negative polarity reference for a port automatically defaults to the closest signal layer (with geometry) or negative layer, above or below the primary terminal.

If the model has a two-terminal excitation, override this default behavior by designating the voltage reference for the port. Typical applications include referencing one signal line to another (a differential pair,) or referencing a trace to a finite negative signal, either in-plane (a CPW) or out of plane (a finite negative signal antenna.) Follow these steps to change a port's voltage reference.

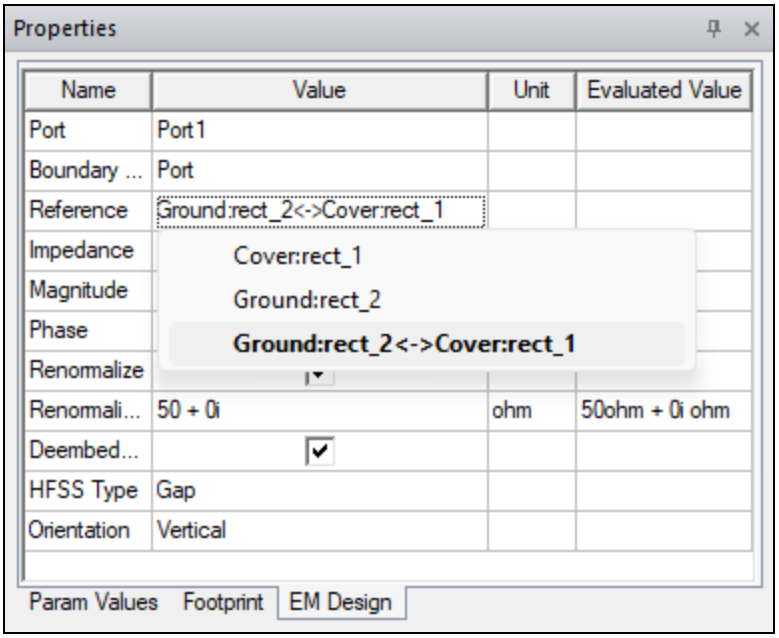
1. Select an edge on a signal layer to mark it as the reference surface.
2. Right-click the excitation in the project tree and select **Add Reference**.

The reference is changed to the edge you selected.

Selecting a Dual Reference

If a lumped port abuts two planes (i.e., two possible reference surfaces), users can select both surfaces. With the port selected, navigate to the **Properties** window. Then select the dual reference option from the **Reference** drop-down menu (i.e., the dual reference option lists both reference surfaces with a **<->** demarcation 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.



Reference to Nearest Negative Signal

When you setup excitations using the Edge Port Definition window, select to select the **Reference** option and specify the nearest negative signal. This option explicitly forces the reference of a port to be located on the nearest negative signal plane, directly below (or above) the port. A vertical trace connecting the port to its reference is added to the structure.

The **Reference** to a nearest negative signal option should be used when the port is over one-tenth of a wavelength on the negative signal plane. If the port is closer than one-tenth of a wavelength, referencing to the nearest negative signal is inappropriate.

> **Warning:**
>
> In a stripline environment, the additional vertical current that results from using the **Reference** option to the nearest negative signal may excite the parallel plate mode — which could create inaccurate results.

Configuring Multiple Ports

It is possible to configure multiple HFSS ports at once. All port attributes exist as HFSS 3D Layout properties. The **Properties** window is populated with a union of properties for the active selection set. To edit many ports at once, select multiple ports to populate the **Properties** window. Finer selection is available by:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

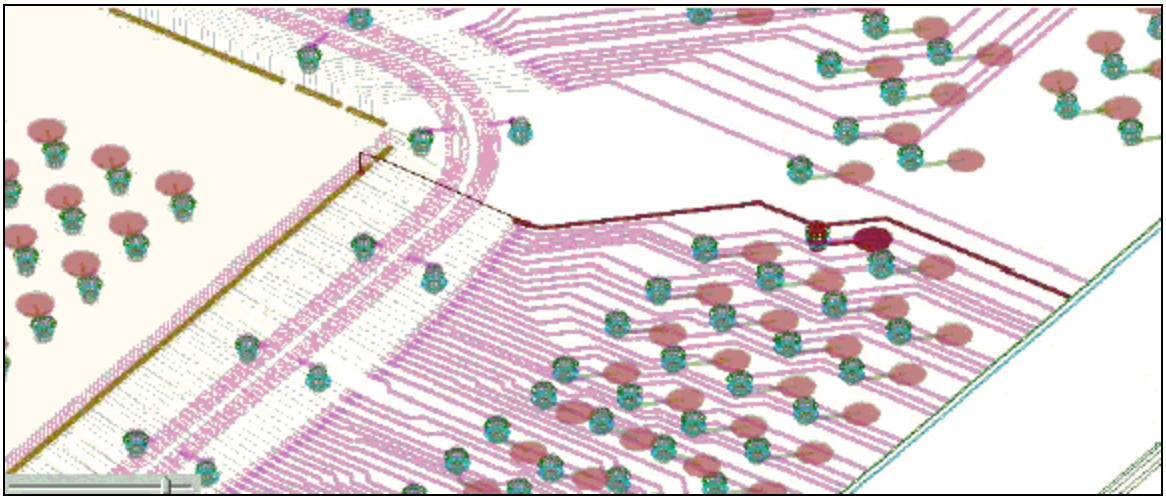
- Editing layer selectability using the **Layout** tab or **Layers** window.
- Using the **List** window

Validating Connectivity and Return Paths

Users can select and view objects that are physically connected to a component in a geometry. The objects can be viewed in the **Design List** and **Editor Layout** window, which can in turn be resized to zoom in on just the objects selected. This workflow allows the user to trace and highlight connected paths across layers in a design in order to validate connectivity and return paths. And then, within this feature, examine complex geometry through selection, net highlighting, transparency, Z-scaling, and cross-sectioning in the 3D view.

To select objects physically connected to a specific object:

1. Select the pertinent objects.
2. Right-click and select **Nets** > **Select Physically Connected**. Alternatively, click **Layout** > **Nets** > **Select Physically Connected** in the menu bar.All connected objects are selected.



To view connected objects in the Design List and the Layout window:

3. Right-click on one of the objects and select **List Selection**. The Design List opens with only its Layout tab.
4. From the window's list, select the row of every object selected in the **Editor Layout** window. Click **Shift** and **Ctrl** if appropriate.
5. Click **Fit Selected**. The Layout window zooms in on the 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.

Note:

Not all geometry types work with the **Select Physically Connected** command. Connections through solderballs, planar bondwires, and very simple instances of 3D-extrusion are supported. Bondwires with 3D geometry and most 3D-extrusions cannot be tracked.

Exposing Horizontal Lumped Ports in Hierarchy

- If a top-level port exposes a port in hierarchy, and that port has an explicit horizontal reference, the **Use Hierarchy Reference** property appears. If this option is selected, the original reference geometry within hierarchy (i.e., in the sub-circuit) is used.

Note:

Enabling the reference in hierarchy results in the removal of the properties: **Reference**, **PortSolver**, **Ignore Reference**, and **Pad port layer**.

Name	Value
Port	Port3
Boundary Type	Port
Use Hierarchy Reference	☑
Impedance	50
Magnitude	1
Phase	0
PostProcess	☑
Renormalize	50 + 0i
Deembed	0

Port Settings for Post Processing

When you specify an excitation for an HFSS 3D Layout structure, you can configure settings for post-processing that are specific to HFSS 3D Layout. These settings are:

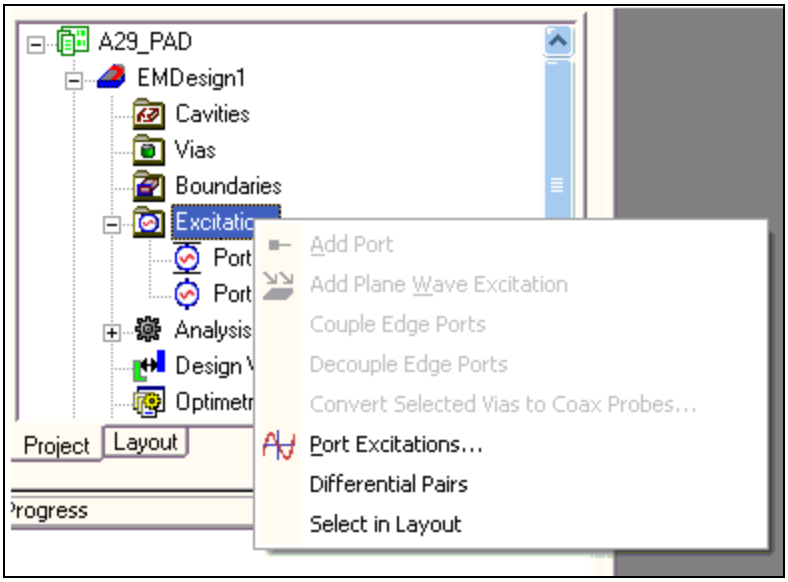
- Edit Port Excitations
- Renormalizing S-Matrices
- Deembedding S-Matrices

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

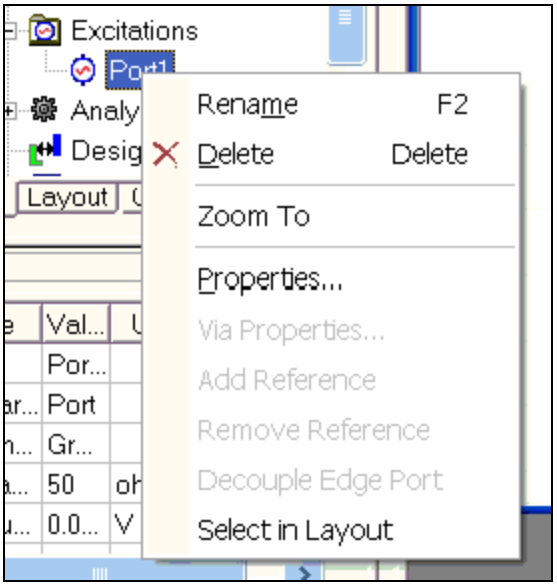
- Varying the Port Excitation
- Differential Pairs

Edit Port Excitations

Many of the excitation features described in the following sections can be accessed by right-clicking the **Excitations** folder in the **Project Manager** window.

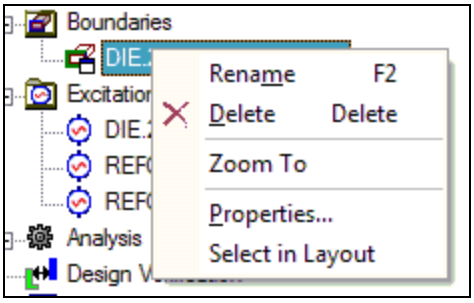


Or can be accessed by right-clicking **Excitations** items and **Boundaries** items.

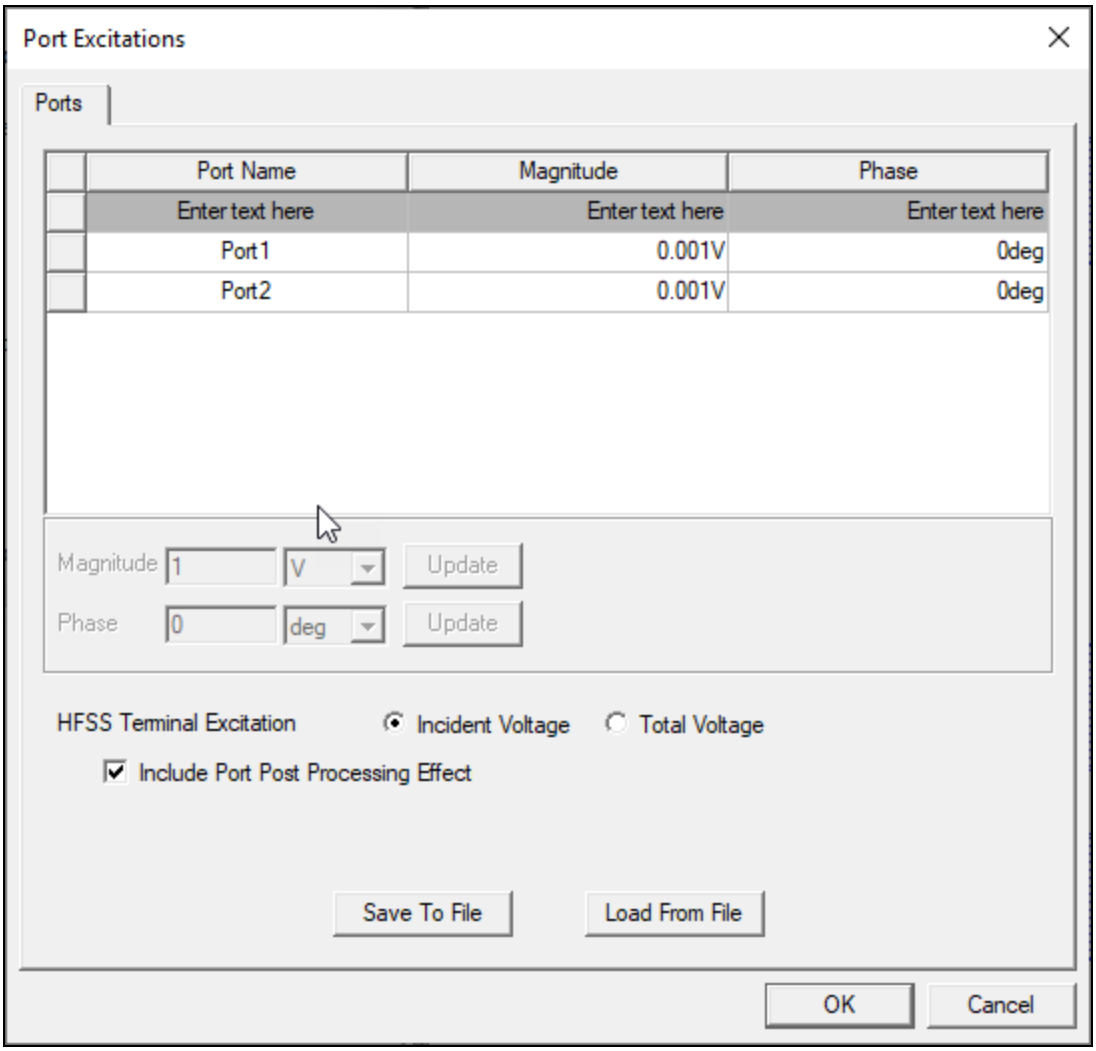


AND

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

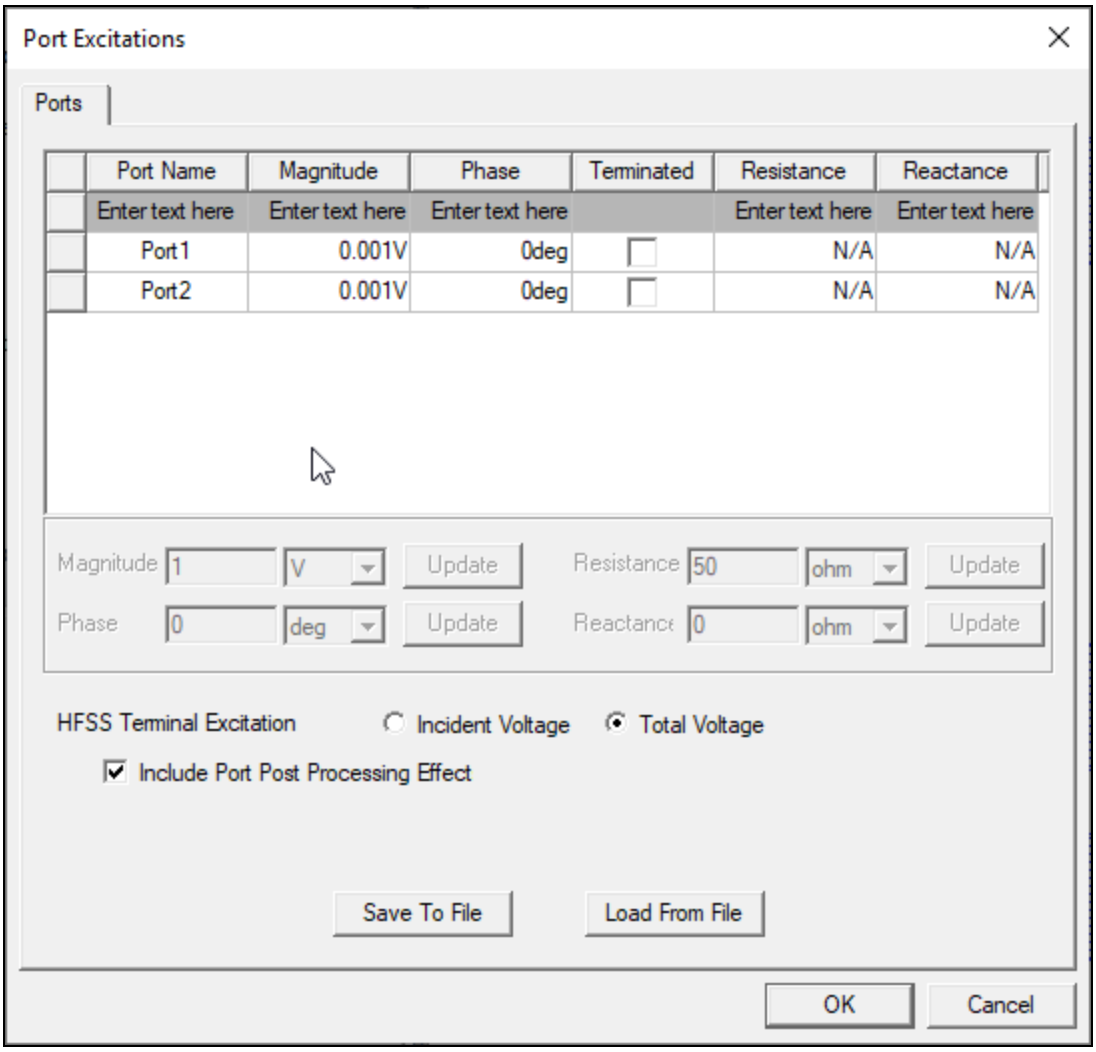


Select **Port Excitations** to open the **Port Excitations** window:



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 this window to modify port excitations, including post-processing and renormalization settings. For HFSS Terminal solutions, select a terminal excitation type of **Incident Voltage** or **Total Voltage**. Selecting **Total Voltage** adds more columns to the table.



You can scale the sources individually through the source table or import a list of the port excitations and their settings from a text or CSV file through **Load From File**. This feature can help for projects with many ports. Follow these steps to directly edit the ports table.

1. Select ports and their corresponding settings fields activate beneath the table.
    a. Use the first row of the table to filter the ports to find the ones 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.

2. From the **Magnitude** field, enter the magnitude you want. Design variables can be used.

> **Note:**
>
> You may not enter a negative voltage. To obtain the equivalent of a negative magnitude, add or subtract 180 degrees on the phase value.
>
> If you use a design variable as a scaling factor note that solutions are invalidated if the variable is changed.
>
> At least one port should be excited (non-zero). If all ports are set to zero, a warning appears, but the values do go through.

3. From the **Phase** field, enter the new phase for the port. The phase of the source is changed by the value that you enter.
4. Optionally, if your solution type is driven terminal, you may specify a complex reference impedance:
    a. For the selected terminal, select **Terminated**. This turns off the values to the left of the check box, and enables the **Resistance** and **Reactance** fields.
    b. Enter the **Resistance** and the **Reactance** and select the units. Ohms is the default.
5. Optionally, click a check box to **Include Post Processing Effects**. The post-processing effect has no impact on the impedance of a terminated port, but the post processing operations (renormalization and/or deembedding) affects the plotted fields.
6. To saves a list of the port excitations in the grid and their settings in a text or CSV file, click **Save to File**.
7. Click **OK**.

> **Note:**
>
> Renormalization is ignored if it is set to zero, but deembedding is still honored. The following warning message is produced for all ports with a zero post-processing renormalization impedance: *Zero impedance on port '<arg1>' is ignored; renormalization is skipped for this port*.

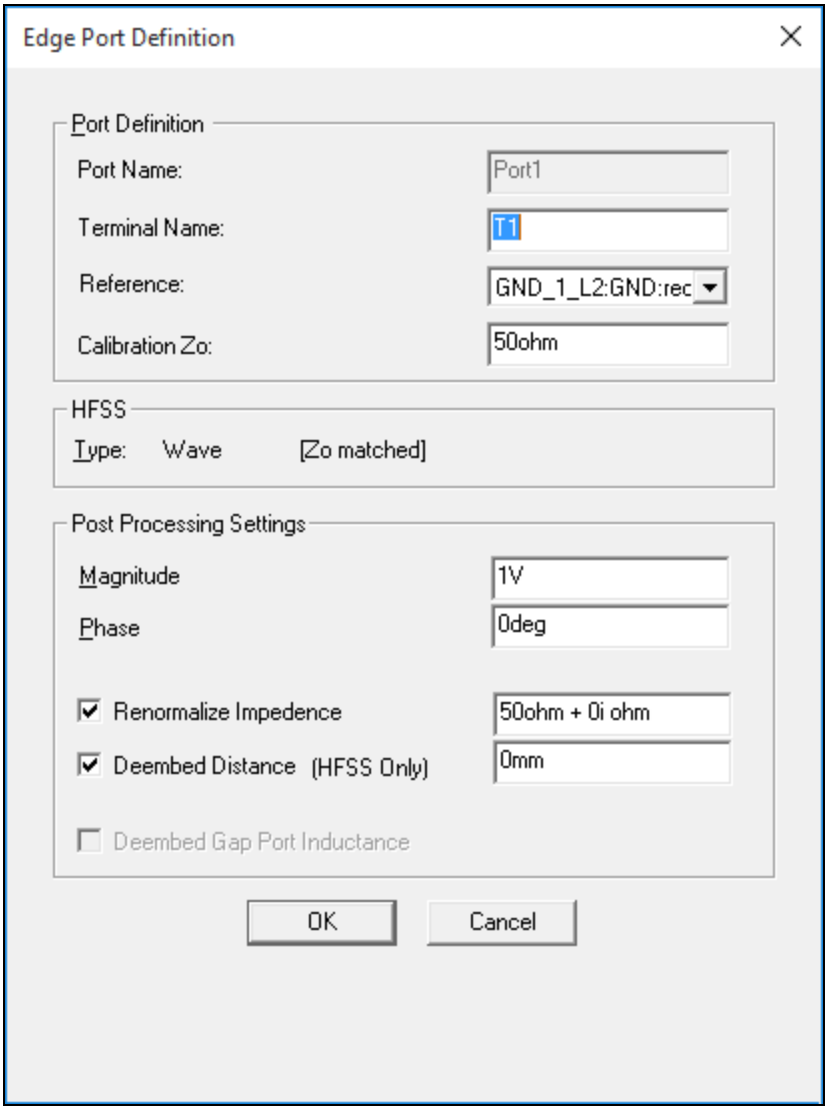
Renormalizing S-Matrices

To renormalize an S-matrix to a specific port impedance:

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]** > **Excitations**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Double-click an appropriate edge port to open the **Edge Port Definition** window.



3. Ensure **Post Process Port** is selected (default).
4. Select **Renormalize Impedence** and set the amount to a non-zero value. The default impedance for re-normalization of each port is 50 ohms.

- If the impedance is a complex quantity, type the real part of the impedance to the left of a plus sign (**+**) and the imaginary part of the impedance followed by **i** to the right of the plus sign (e.g., enter **50 + 10i**).

4. Enter the length of the transmission line to add in the **Deembed Distance** field.
5. Click **OK** to renormalize the port to the new impedance.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

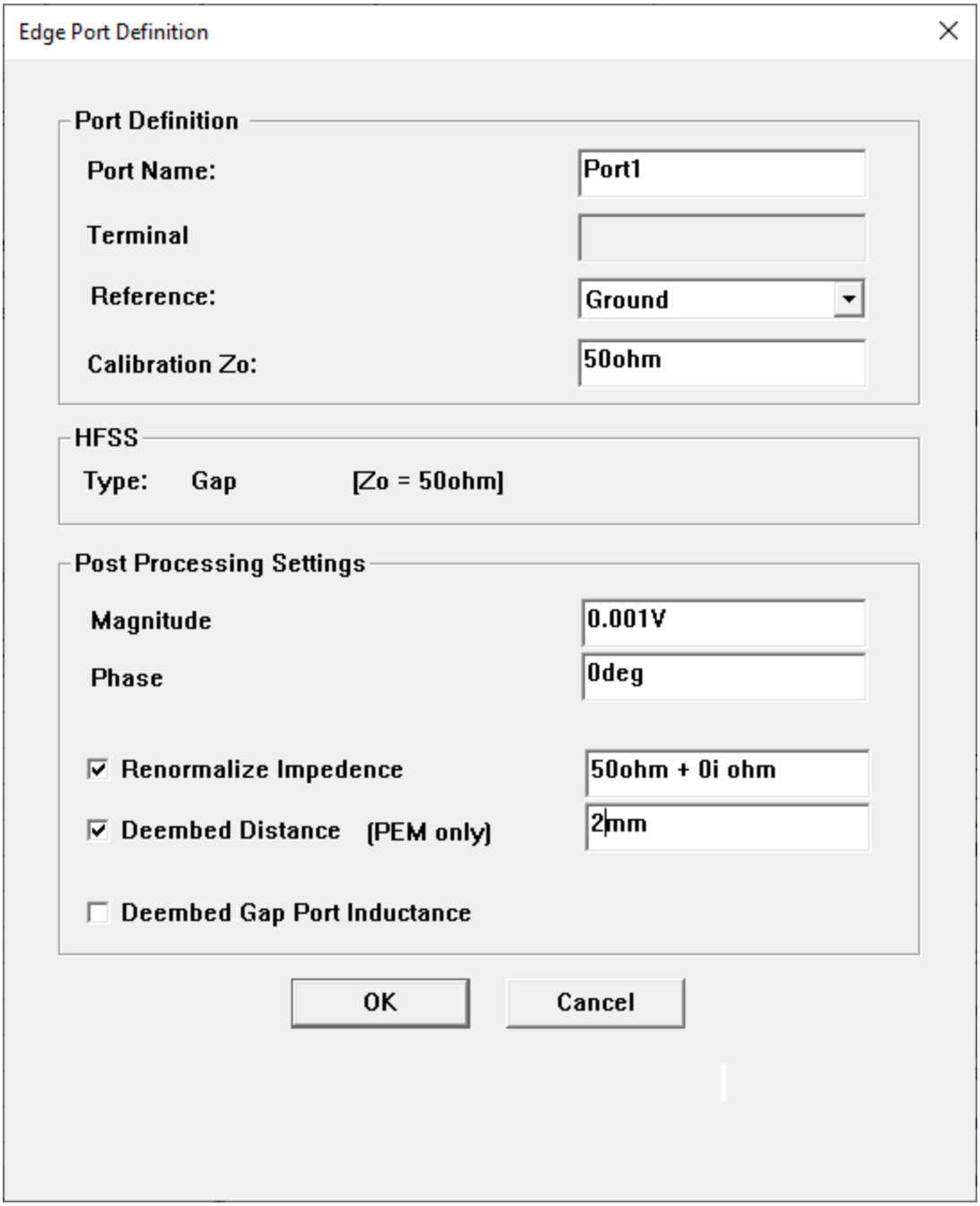
You do not need to re-run a simulation in order to renormalize a port. The 2D Reporter automatically updates its reports to reflect the renormalized S-matrix.

Deembedding S-Matrices

Follow these steps to compute a deembedded or phase-rotated S-matrix.

1. From the **Project Manager** window, expand the **Project Tree** and the **Excitations** folder. Then double-click a port to open the **Edge Port 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.

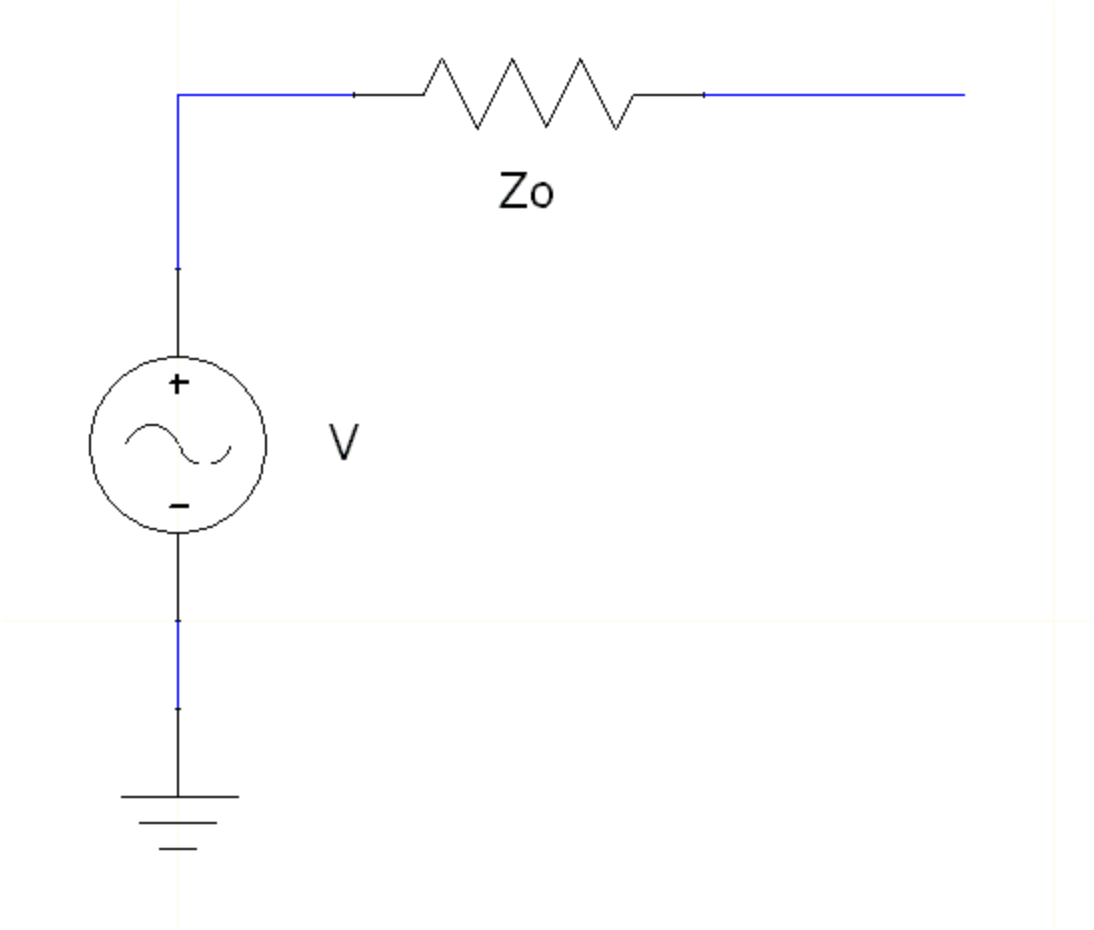


2. Click **Renormalize Impedence** and set the amount to a non-zero value.
3. Enter the length of the new transmission line in the **Deembed Distance** field.
4. Click **OK** to assign that length to the selected port. The 2D Reporter automatically updates its reports to reflect the deembedded S-matrix.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Varying the Port Excitation

You can vary the voltage applied to a model by scaling its magnitude and modifying its phase in the **Edge Port Definition** window. A schematic of an excitation is shown in the following diagram. The voltage source model is in series with an impedance called Calibration Zo that is equal to the characteristic impedance of the port. Hence, renormalizing the port impedance change the port excitation model.

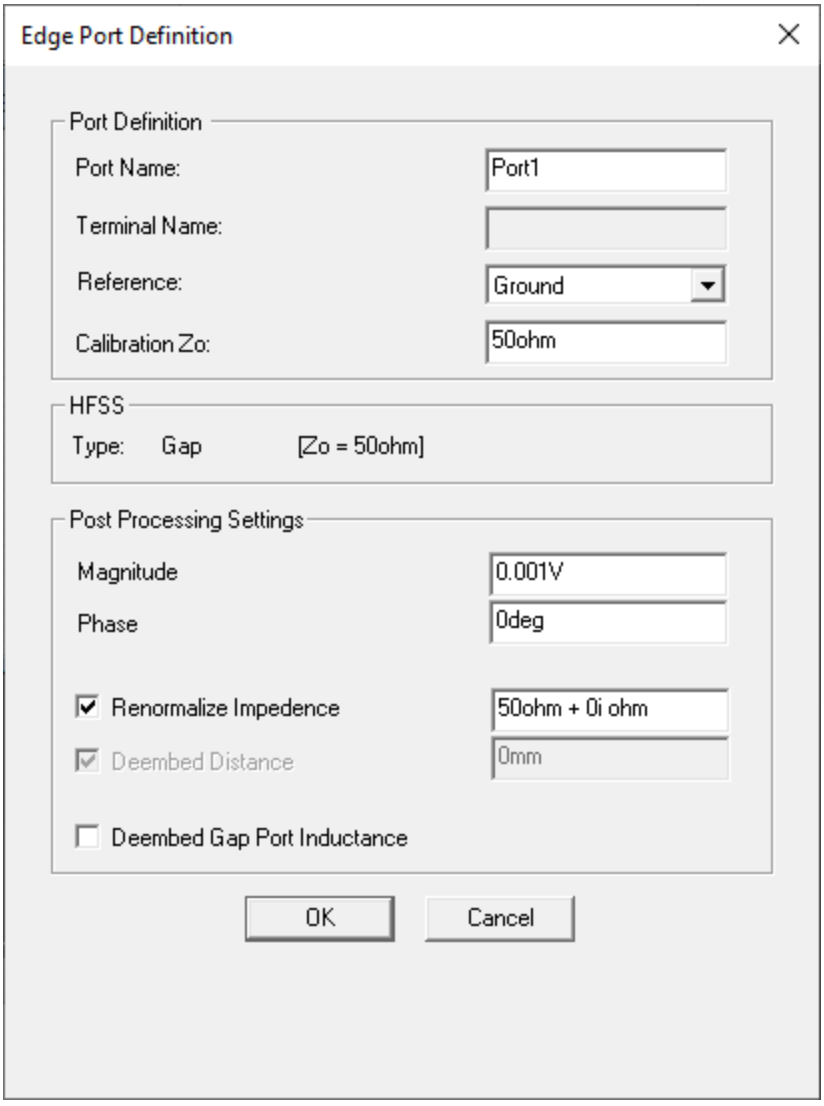


Zo is equivalent to the Full Port Impedance of the port. It is user-defined with the format "<real_part> + <complex_part>*i ohm" (e.g., example 50ohm + (-5ohm) * 1i). You can assign a variable as these values, (e.g., " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (e.g., pwl (ds1,freq) + (pwl(ds2,freq)) * 1i). The real part must be positive and the complex part non-zero if the resistance is zero. See lumped port theory for details.

To vary the applied voltage at a port:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Double-click the edge port in the project tree to open the **Edge Port Definition** window.



2. For the ignore reference option:
- If not selected, the port and the reference plane are physically connected. For most cases, this has better high frequency performance.
- If selected, the port and the reference plane are not connected. This is typically the better option for a stripline environment since a vertical current between two infinite ground planes can excite a parallel plate mode.
3. For the port solver option:
- If selected, the Z_0 and gamma are calculated and the port is calibrated.
- If not selected, Z_0 is user-defined (default is 50) and the port is not calibrated.
4. From the **Magnitude** field, enter the magnitude of the voltage applied at the port.

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, you may use the default value of 1 volt. This specifies the solution's current is scaled so that the excitation source is 1 volt. To view the solution at a different excitation level, enter a different positive value for **Magnitude**. Only modes with non-zero magnitudes are used in post-processing.

5. From the **Phase** field, enter a phase of the voltage applied at the port.
6. Click **OK**.

Differential Pairs

A differential pair signal resides on two conducting traces in the presence of a ground plane. To form the differential pair, the transmitter launches a wave that carries equal but opposite voltages on the two traces (e.g., one trace is +1 volt and the other -1 volt). When two traces are routed in close proximity to each other, the waves traveling on those traces exhibits similar amounts of system noise. A differential pair is an effective way to remove noise from a signal because when a receiver processes a differential signal it subtracts the voltages on the two traces. This can dramatically reduce the amount of noise in the differential signal compared to if the traces are driven individually (single-ended signals). In HFSS, a differential pair can be defined for two terminals residing in the same wave port or for terminals that reside in two different wave ports or lumped ports. From the case of wave ports, matched differential pairs can be defined by not renormalizing the wave ports involved in the differential pair definitions.

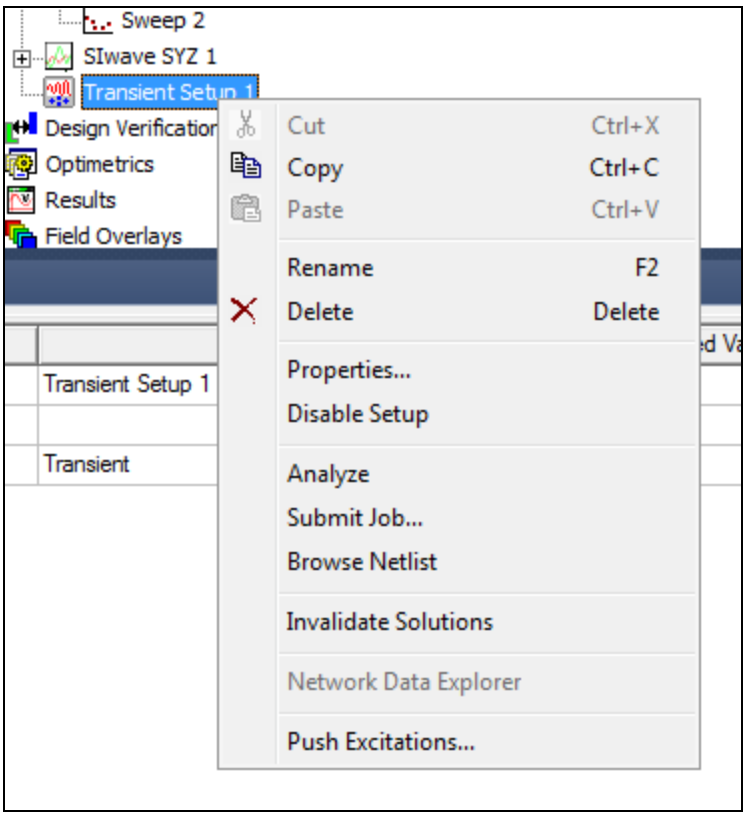
Pushing Excitations

Push Excitations combines Circuit and HFSS 3D Layout features to extract excitations from a Circuit simulation to use in the post-processing of an HFSS 3D Layout design (e.g., if you have a model that you extracted with a field solver, you can import that model as a “black box” into Circuit, solve its circuit, then use the solution to calculate the model’s excitation vector. The vector is pushed back to HFSS 3D Layout to solve for the model’s field distributions.). Excitations can be pushed to HFSS 3D Layout ports, to dynamic links, and to subdesigns. Excitations can also be pushed if they have a Thevenin Equivalent.

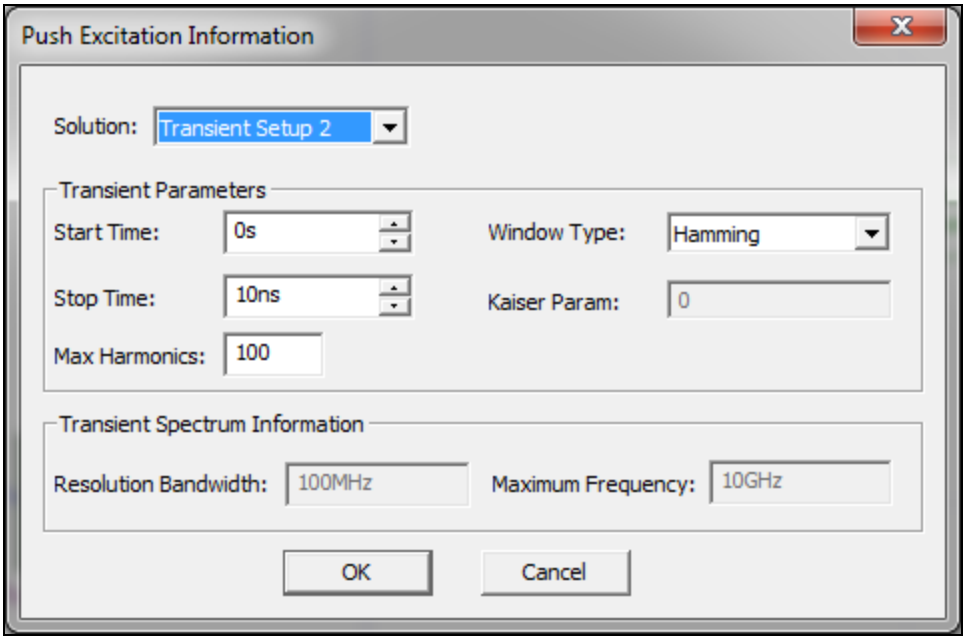
Push Excitations to HFSS 3D Layout Ports

Excitations are calculated at the 3D Layout ports and are pushed in the same design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

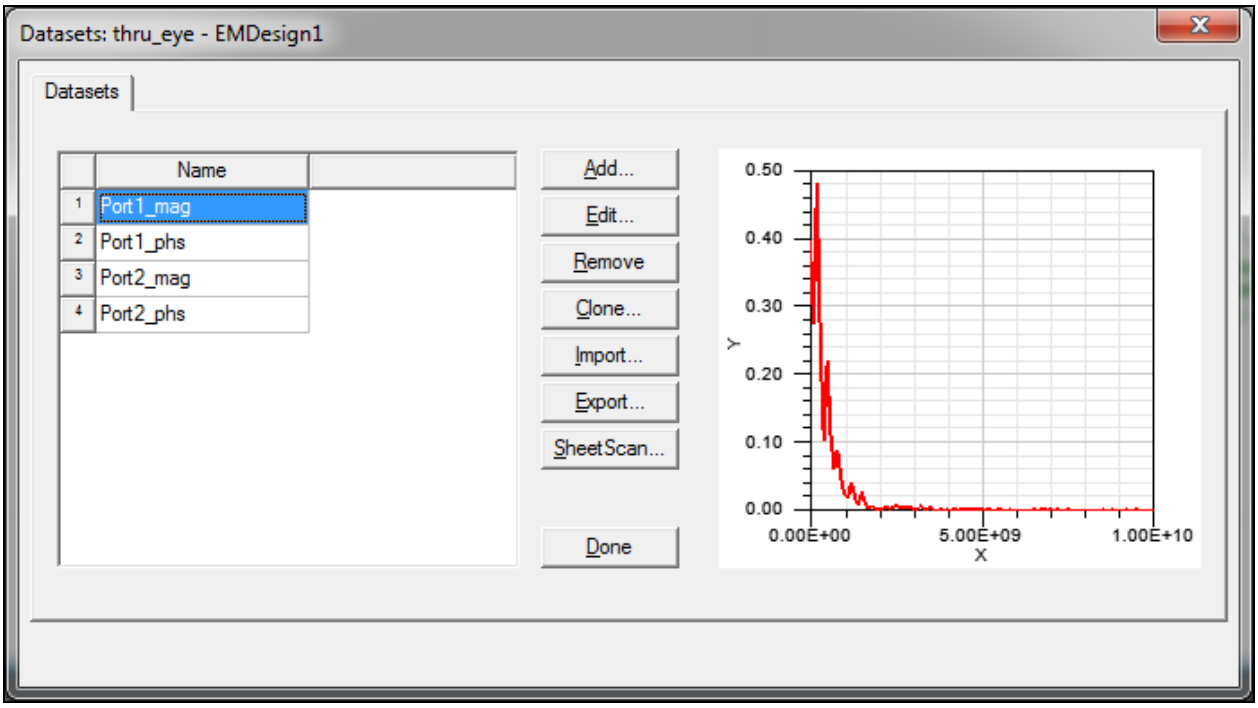


Users can right-click a Nexxim solution and select **Push Excitations** to open the **Push Excitations Information** window.



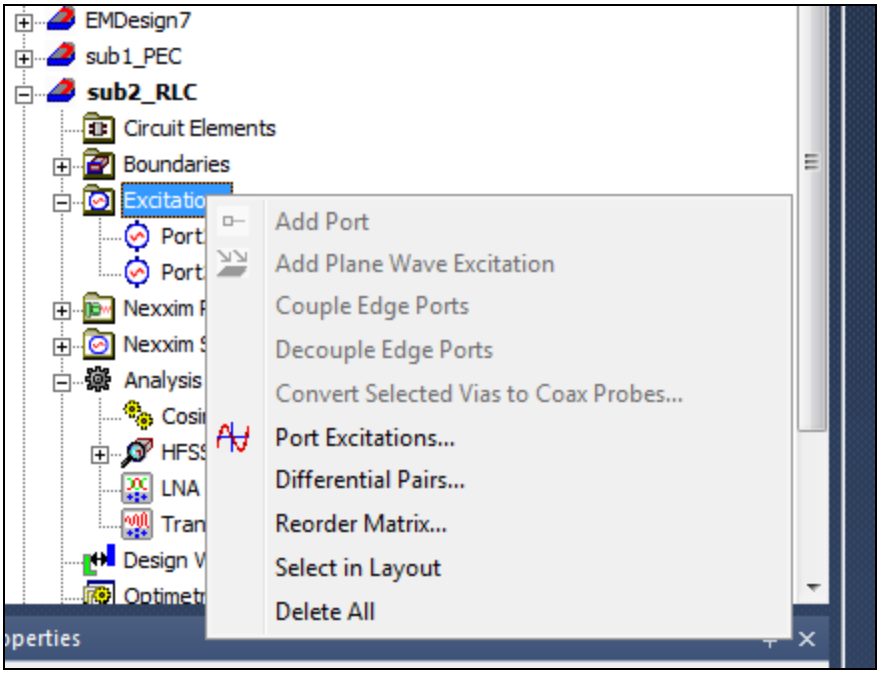
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data sets are created using these port excitations.

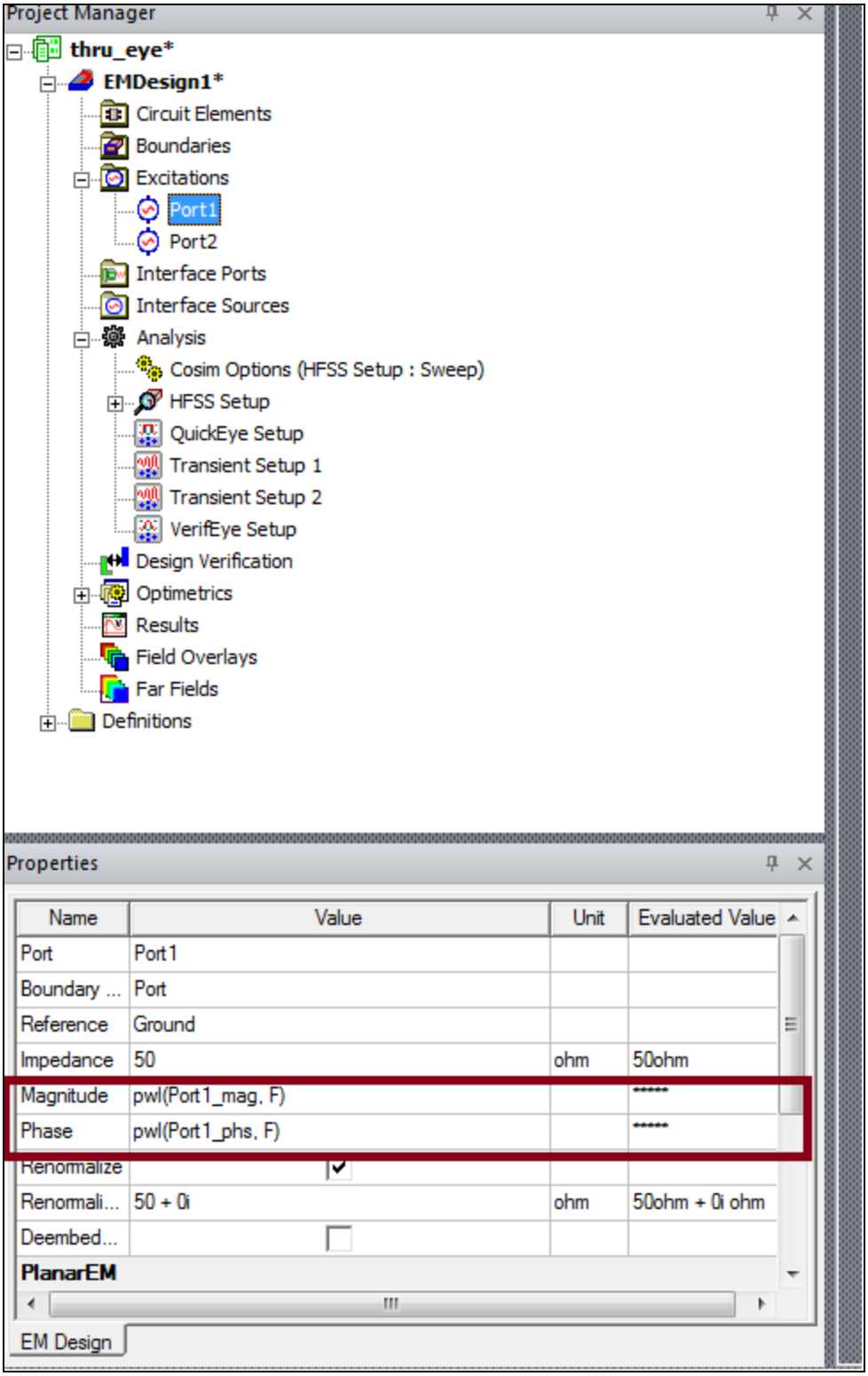


Voltages are extracted and port-excitations are calculated. The **Port Excitations** option in the shortcut menu that is accessed on the **Excitations** folder also allows users to edit excitations at 3D Layout ports.

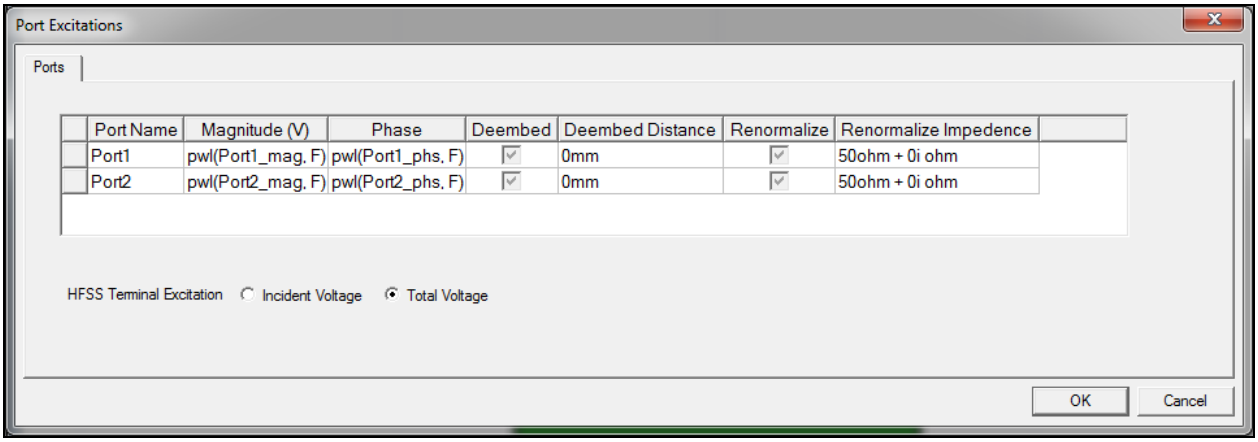
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



The excitation calculated for each port is also available on the **Port Excitations** window and from Port Properties. Far Field plots and Field Overlays are updated with the new excitations.

Push Excitations to Dynamic Links

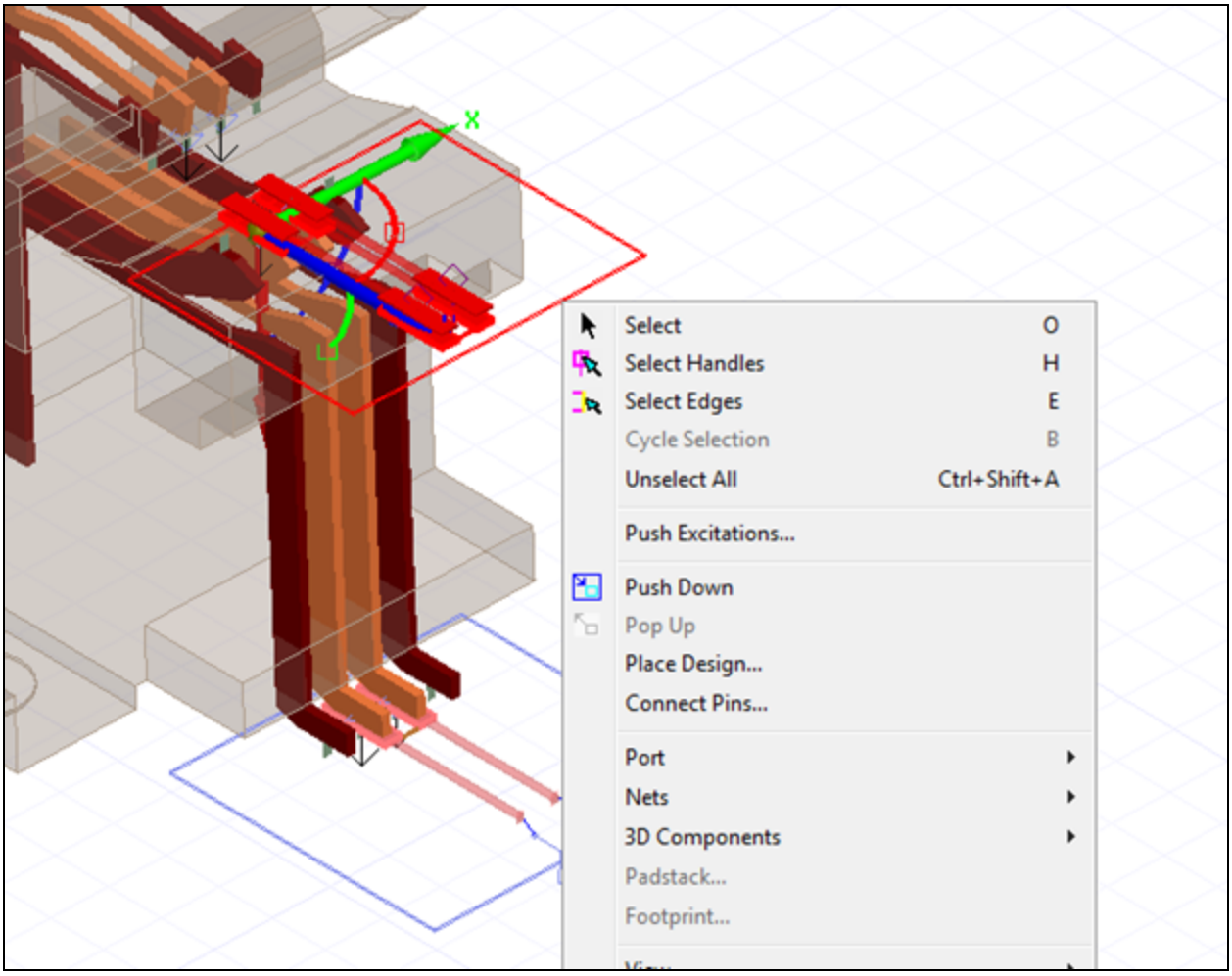
Select a dynamic link in the Layout Editor. Then right-click to see the **Push Excitations** option. The option is only available if there are Nexxim solutions to be pushed. Excitations that are pushed can be viewed in HFSS design.

Push Excitations to Sub-designs

To push excitations select a sub-design in Layout Editor and right-click to see the **Push Excitations** option. The option is only available if there are Nexxim solutions available to push.

Excitations that are pushed can be viewed by going to the sub-design and updating or plotting Far Field and New Field plots or plots with currents. The **Excitations** area in the sub-design also provides feedback if excitations have been pushed.

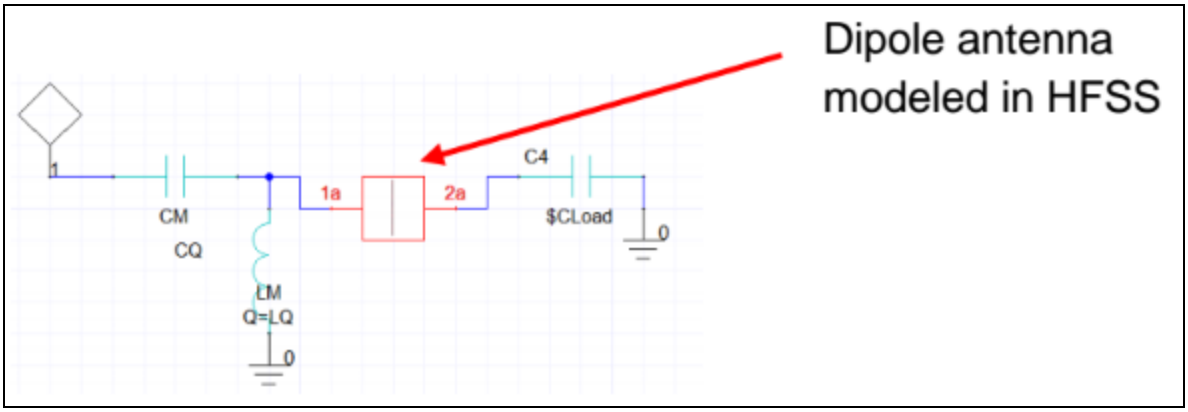
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



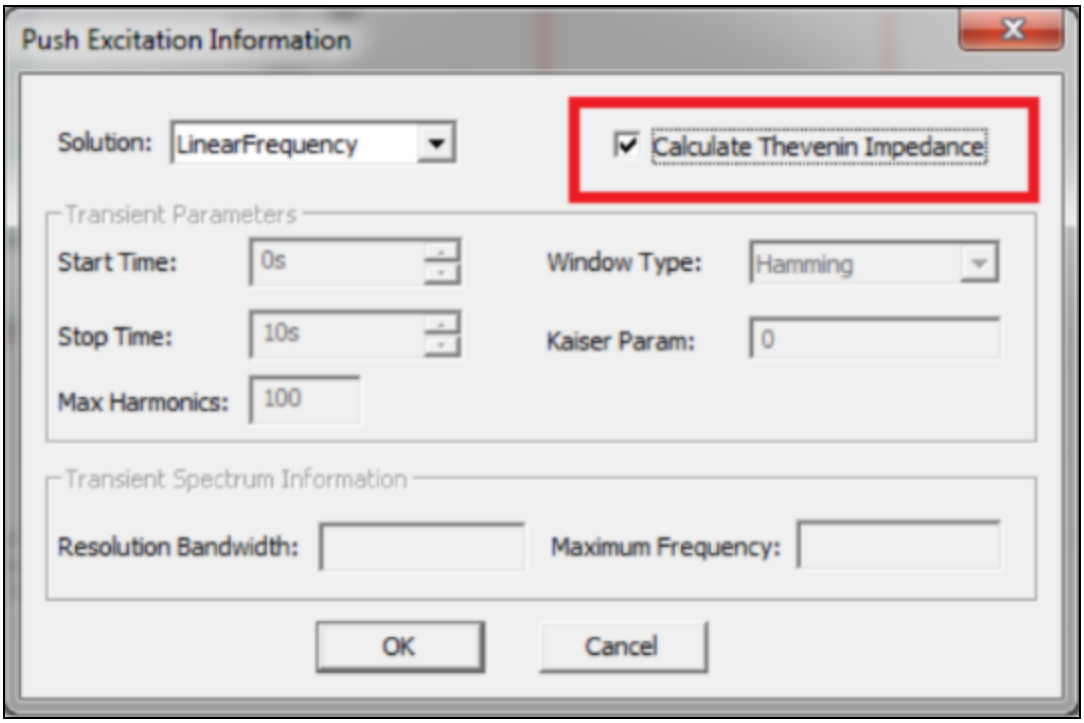
Push Excitations with Thevenin Equivalent

A Thevenin Equivalent is the efficiency and realized gain of an antenna in a circuit. The Thevenin Equivalent calculation is supported only with a LNA solution. To capture the loss of any intermediate circuitry (such as a matching circuit) the design needs to push incident power at the Circuit ports and push the Thevenin Equivalent at the HFSS ports. These values can be computed by HFSS.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A push excitation with a Thevenin Equivalent is configured through a dynamic link, with the complex impedance defined in the Circuit Tool and with the antenna defined as an HFSS dynamic-link sub-circuit.



- From the **Push Excitation Information** window, the **Calculate Thevenin Impedance** check box is cleared by default.
- Thevenin impedance values are calculated per pin and are pushed to HFSS when the check-box is selected. The impedance is *0 ohms* for shorted pins and is *inf ohms* for open pins. These values are functions of frequency.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For an HFSS discrete sweep, the push excitations, available power, and Thevenin impedances are at HFSS sweep frequencies. For all other cases, pushed data corresponds to frequency points on the Nexxim solution.
- The pushed renormalized port-impedances can be examined at the port properties. Pushed source-settings can be viewed in the **Edit Post Process Sources** window.
- For pushing excitations to HFSS 3D Layout, there is a control for **Target Solution**. Its drop-down menu shows adaptive and discrete solutions that have the **Save Fields** option selected by default.

Error Handling

- If errors are encountered while calculating the impedances, a message are posted and excitations are not pushed.
- If invalid excitation values are pushed to HFSS, error messages are posted and the pushed values are not applied. Examples of invalid excitation values include:

  -inf impedance values

  <=0 system power

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.

10 - Solution Setup in HFSS 3D Layout

After drawing a model and defining its excitations, you must specify how HFSS 3D Layout computes the solution. You can define one or more sets of solution parameters to solve. Each solution set appears as one *setup* in the **Project Manager** window. Choose multiple types of analysis setup for an HFSS 3D Layout depending on the selected solver. Time Domain Reflectometer (TDR) plots are available for LNA and imported solutions in HFSS 3D Layout. After a solution is set up, you can modify it as needed. High Performance Computing (HPC) options that allow you to speed up processing.

> **Note:**
>
> Adding an HFSS analysis setup, editing frequency sweeps, and analyzing the setup are functions performed the same in both **General** and IC modes, although the options available in General mode differ from the options available in IC mode.

Adding a Solution Setup for HFSS 3D Layout

Each set of unsolved solution parameters is listed as a setup under **Analysis** in the **Project Tree**.

To add a new solution setup to a project using initial meshing tools:

1. Do one of the following:
   - From the **HFSS 3D Layout** menu, open **Solution Setup**.
   - From the **Project Manager** window, expand the **Project Tree** and right-click **Analysis**.
2. To open the **Solution Setup** window, select a solution type from the menu.

The easiest way to set up a solution is to accept the **Fixed Mesh** and **Frequency** defaults that appear in the HFSS 3D Layout or HFSS setup window, then click **OK**, which automatically opens the **Sweep** window. Click **OK** to accept the sweep with the suggested defaults. Alternately, change the default values or add more setups by referring to one or more of the following topics.

Adding an HFSS Solution Setup (General Mode and IC Mode)

For HFSS Terminal Solution types, select two HFSS solution setups: auto and advanced. The auto setup offers a simplified interface balanced between accuracy and speed, without many configurations. Advanced provides greater control of the solution setup in exchange for more complex configurations. For more information about HFSS solutions, see in *An Introduction to HFSS*.

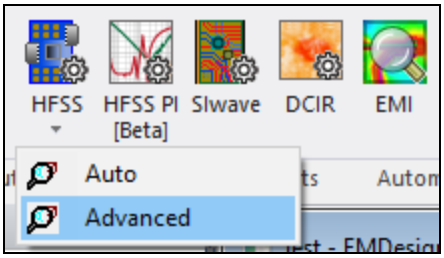
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define bounding computational regions, see HFSS 3D Model Extents.

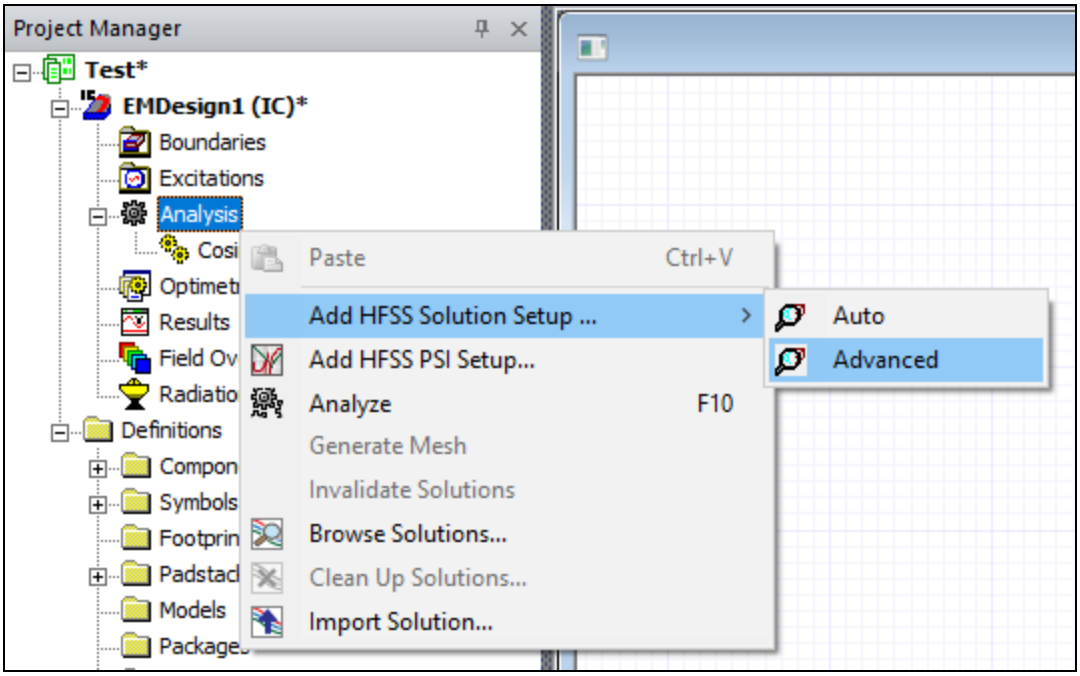
> **Note:**
>
> For HFSS setups, there is an extra solution “Adaptive Pass” to plot S-parameter data for all adaptive solutions. The primary sweep is “pass” which can be swept from 1 to the number of adaptive passes in the setup.

Follow these steps to add an **HFSS Solution** setup to a project in **General** or IC mode.

1. Open the **Setup** window by doing one of the following:
   - From the **Simulation** ribbon tab, click **HFSS** > **Advanced**.



   - Right-click **Analysis** in the **Project Manager** window and select **Add HFSS Solution Setup** > **Advanced**.

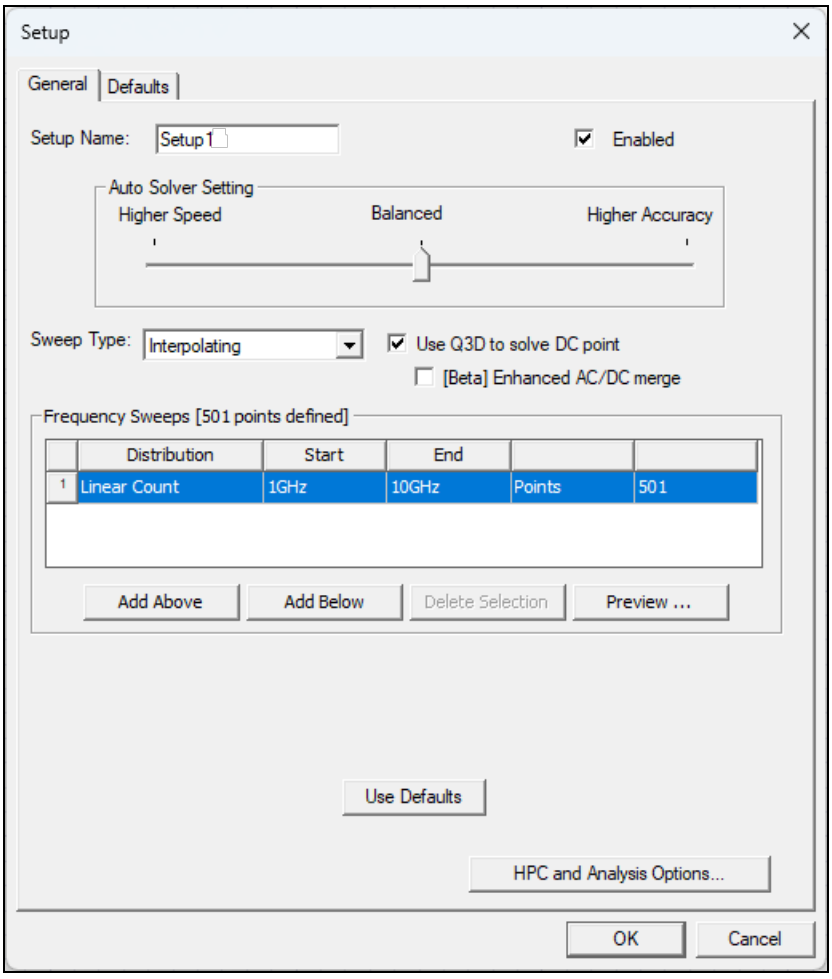


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Do one of the following:
   - Click **HFSS 3D Layout** > **Solution Setup** > **Add HFSS Solution Setup**.
   - From the **Project Manager** window, right-click **Analysis** > **Add HFSS Solution Setup**.
2. Click Auto or Advanced to open the **HFSS Setup** window.

Adding an Auto HFSS Solution Setup

Auto setup includes a simplified window with an **Auto Solver Setting** slide bar from **Higher Speed** (i.e., lowered accuracy to achieve optimum speed) to **Balanced** (i.e., a balance between speed and accuracy) to **Higher Accuracy** (i.e., optimal accuracy at the lowest speed).



The auto setup includes a single sweep, to edit the **Distribution**, **Start**, and **End**. The design must include one or more ports. The **Sweep Type** can be **Discrete** or **Interpolating**:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Discrete** generates field solutions at specific frequency points in a frequency range.
- **Interpolating** estimates a solution for an entire frequency range using an adaptive rational function interpolation method. An Interpolating fast sweep will result in faster analysis, however the current distribution values cannot be saved.

Check the **Use Q3D to solve DC point** box to invoke the Q3D Extractor to explicitly solve for a DC point. The mesh from HFSS is used for the Q3D extraction, with sources and sinks defined based on HFSS port assignment. Matrix reductions are automated to integrate the Q3D results with HFSS.

> **Note:**
>
> **Use Q3D to solve DC point** provides network data only and cannot be used for fields overlays.

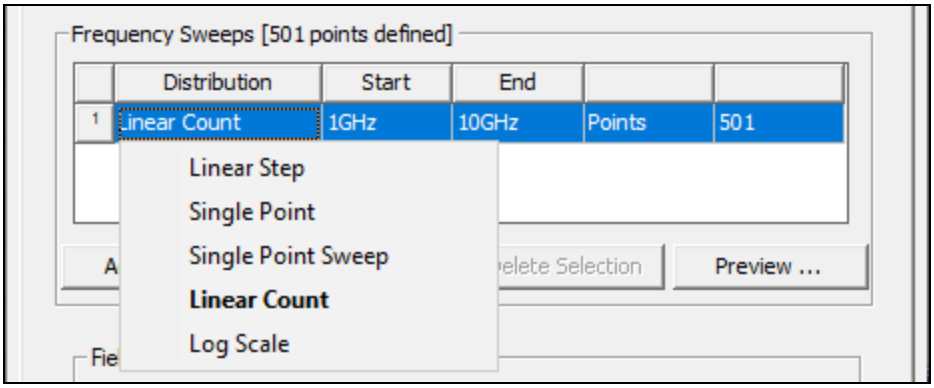
If the **Sweep Type** is **Interpolating** and the **Use Q3D to solve DC point** box is checked, the **Enhanced AC/DC merge** check box activates. Check the box to enforce causality and generate S-parameters at the DC point. By combining the DC results with the frequency-swept AC results, accuracy is improved over the entire simulation bandwidth.

Sweeps can be of the following **Distribution** types:

- Linear Step
- Linear Count
- Log Scale
- Single Point
- Single Point Sweep, which adds a set of 10 single Point sweeps, defaulting from 1 GHz to 10 GHz in increments.

To add more sweeps, including mixed sweep types, click **Add Above** and/or **Add Below**. This feature provides flexibility (e.g., define sweeps with log scale at lower frequencies and linear step at higher frequencies). Frequency sweeps cannot be added to auto setups 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.



To remove a sweep, highlight it and click **Delete**.

Preview displays the sweep(s) as currently defined.

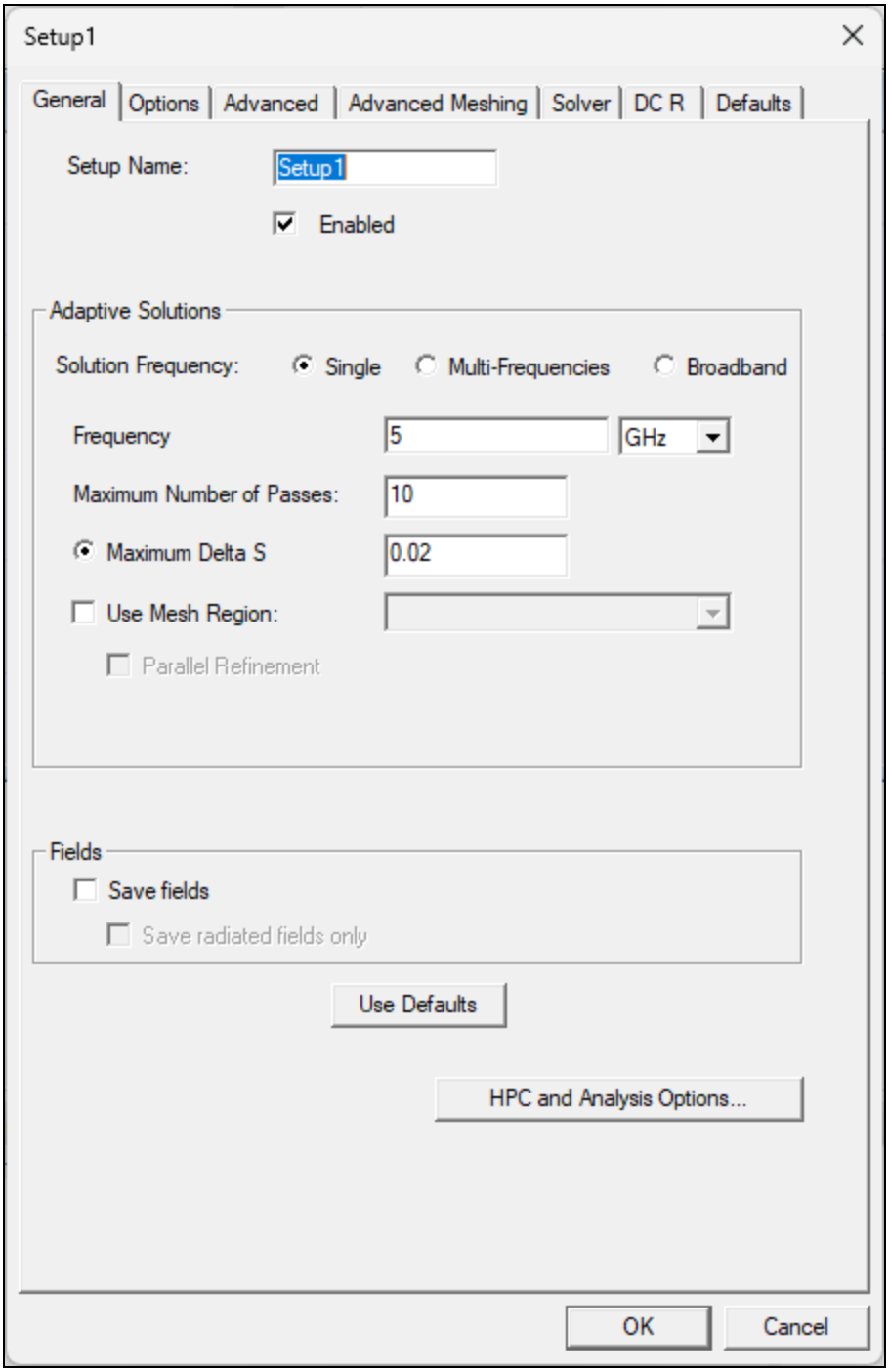
The auto **Solution Setup** window has two tabs:

General	Includes general solution settings. Specify the name, change the slider bar to do different solver settings, select sweep type, set frequency range, and configure **HPC** settings. The selection in the solution setup dictates the behavior of the frequency sweep solve. Changing the selected lists invalidates the last adaptive solution. In a **Discrete** sweep, if a Near Field or Far Field Radiation setup is created, and a list is selected, a warning appears if your selections do not match those selected for the solve.
Defaults	Saves the current settings as the defaults for future solution setups or revert the current settings to HFSS 3D Layout standard settings.

Adding an Advanced HFSS Solution Setup

Click **Advanced** to open the **Setup** window.

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** window is divided into the following tabs:

General General solution settings.

Options Settings for lambda refinement, adaptive analysis and solution options, the Order of Basis setting, and whether to enable the use of solver domains, if

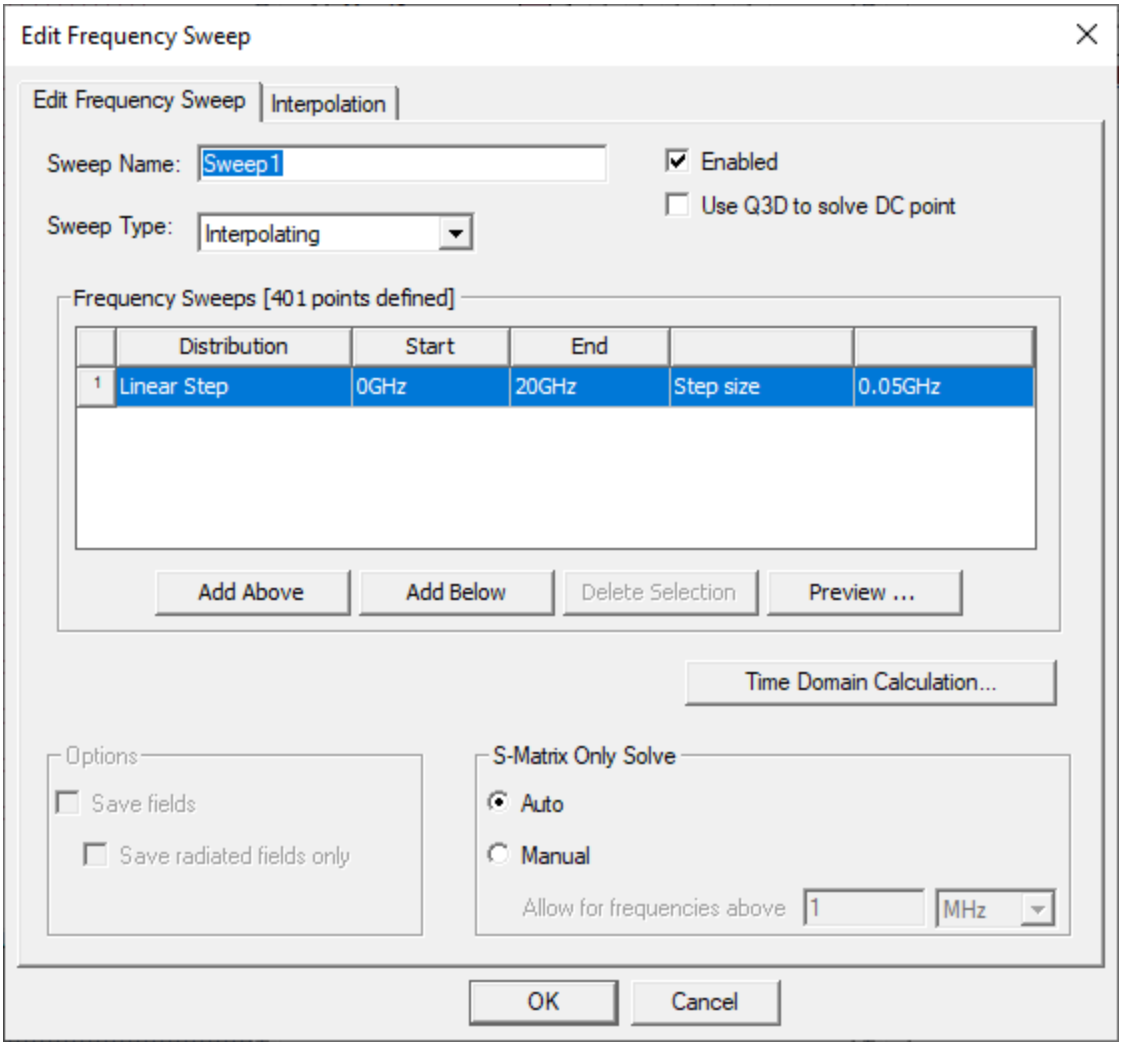
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	available for the solution type.
Advanced	Settings for defeaturing a mesh, which model type to use, and vias.
Advanced Meshing	**(General Mode Only)** Settings for curve approximations and layer alignment.
Solver	Port settings and configurations for modeling.
DC R	Settings for DC Resistance analysis.
Defaults	Saves the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

1. Click the **General** tab.
2. If applicable, clear the **Enabled** check box so the setup does not run automatically when Analyze All is applicable,.
3. Select a **Solution Frequency** and configure it.
4. For an adaptive analysis, enter the Maximum Number of Passes and then specify the remaining adaptive analysis parameters for HFSS.
5. To select or create an analysis configuration, click HPC and Analysis options.
6. To post-process or plot fields, click **Save Fields**.
    - To only analyze the far field, click **Save radiated fields only**.

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 **OK** to close the **Setup** window and open the **Edit Frequency Sweep** window.



8. From the **Edit Frequency Sweep** window, click **OK** to accept the sweep with its default values or change the default values. Refer to Adding a Frequency Sweep for more information.

(IC Mode Only)

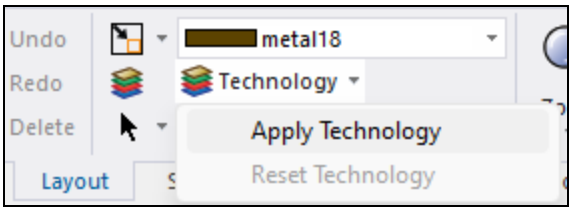
9. If any technology files were imported during the initial setup of the design, follow these steps to apply a technology before simulation:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

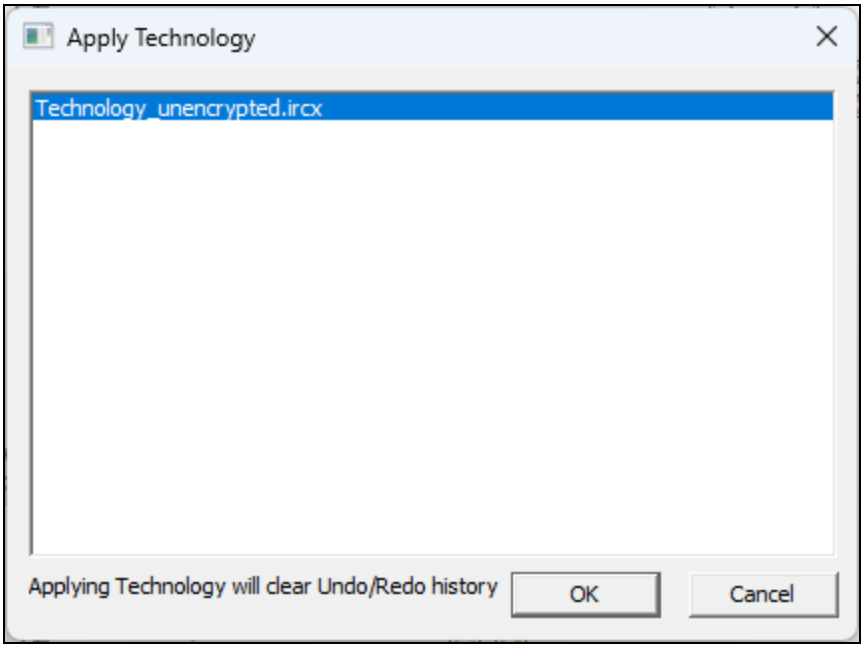
Note:

If technology files were **not** imported during the initial setup of the design, users can add one or more files at any time before simulation. If a technology file **was** applied during setup, the technology's name should already appear in the **Layout** ribbon adjacent to the **Technology** drop-down menu. Users will not need to follow these steps. Refer to Adding a Technology File to a Design.

a. If a technology is available, navigate to the **Layout** ribbon and select **Apply Technology** from the **Technology** drop-down menu.



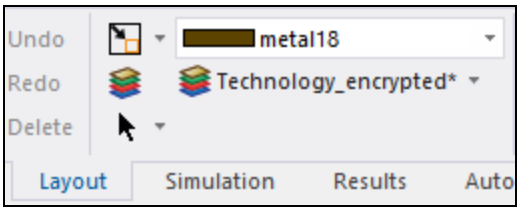
b. Once the **Apply Technology** window opens, select an appropriate technology from the list and click **OK**.



Once the technology is applied to the design, the technology's name appears

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

adjacent to the **Technology** drop-down menu.



10. Once the simulation is prepared, run the analysis to solve the setup. Refer to Running Simulations.

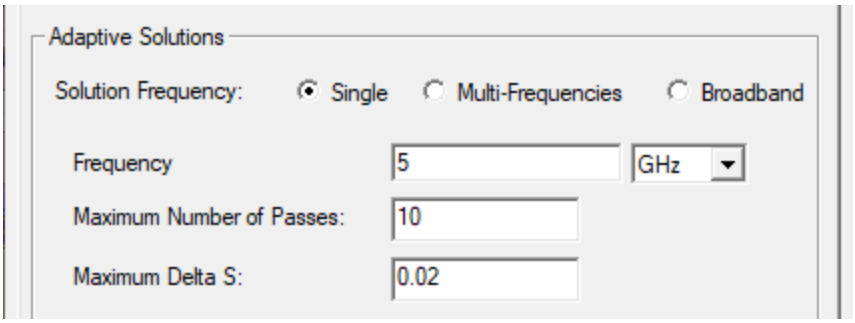
Setting the Solution Frequency

For every Driven solution setup, specify whether to solve a **Single Frequency**, **Multi-Frequency**, or **Broadband**, and provide the **frequency** information and units at which to generate the solution. Each method changes the fields shown in the dialog to those needed for the selection. If you enable HPC, multiple frequencies will be distributed.

If a single frequency sweep is solved, an adaptive analysis is performed only at the solution frequency. If you want to solve over a range of frequencies, define a frequency sweep or a Mult-Frequency or Broadband solve. You must first assign an excitation for the Mult-Frequencies and Broadband options to appear on the setup.

Single Frequency

If you select Single for solution frequency, you specify a value and units with a maximum delta S (or Matrix Convergence), plus the maximum number of passes.



- Enter a value for **Solution Frequency** and select the frequency units from the pull down list.

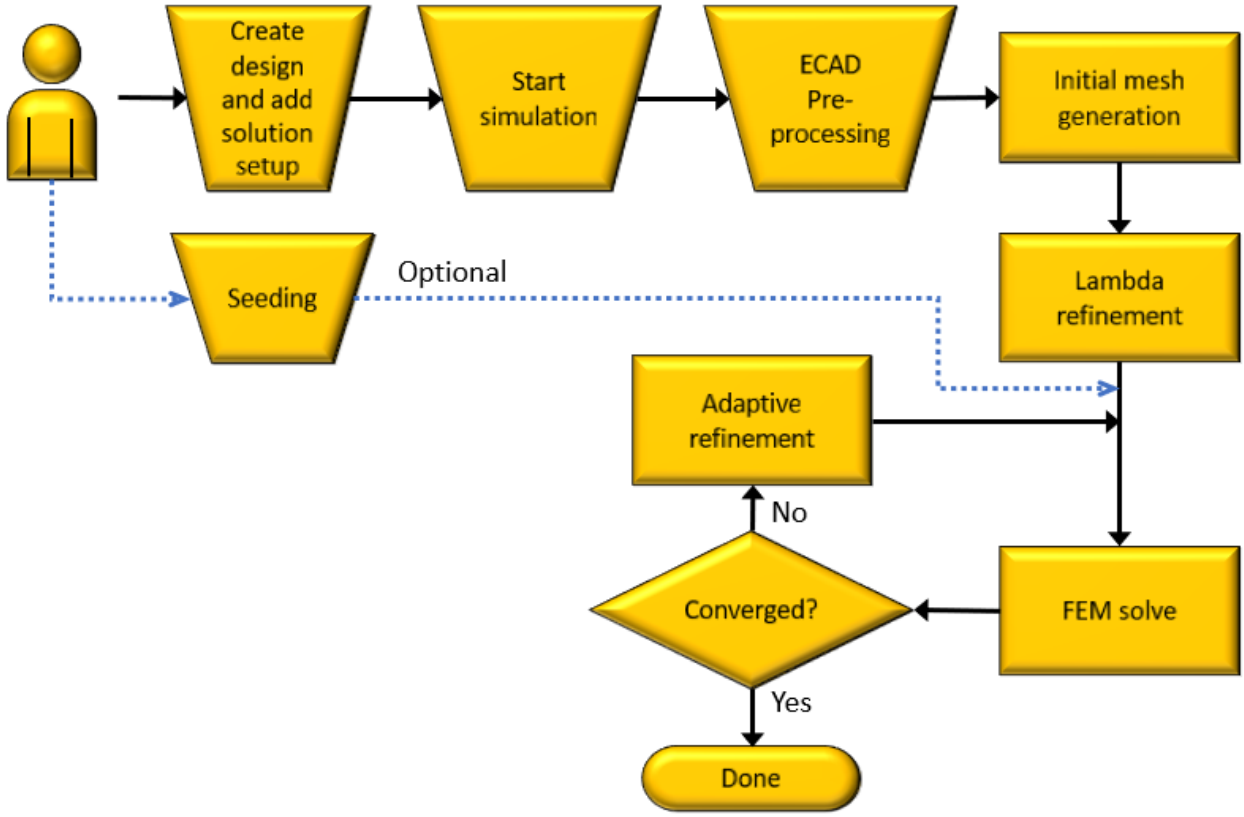
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify the **Maximum Number of Passes** and Maximum Delta S.

> **Note:**
>
> For Fast sweeps, HFSS uses the solution frequency as the center frequency if it is within the frequency range (greater than the start frequency and less than the stop frequency.) Otherwise the middle of the frequency range is used as the center frequency.

Multi-Frequency

If you select **Multi-Frequencies**, you can click **Add** to add additional frequencies, and edit the value, units, and Max. Delta for each frequency. You can also assign existing output variables in these fields.



You can use the **Add** button to specify one frequency at a time.

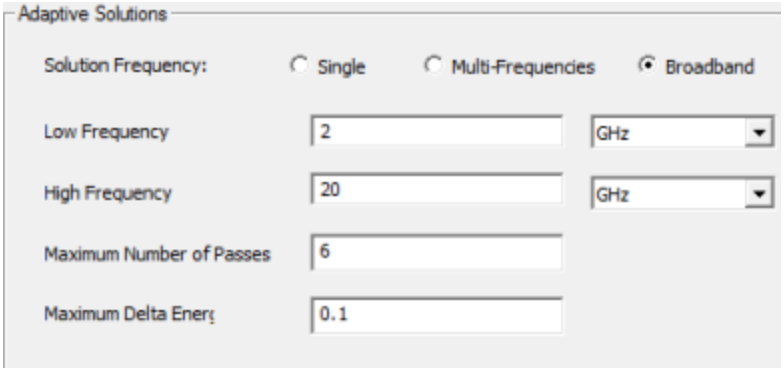
You can use **Remove** to remove any selected items in the Frequency Sweeps table.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

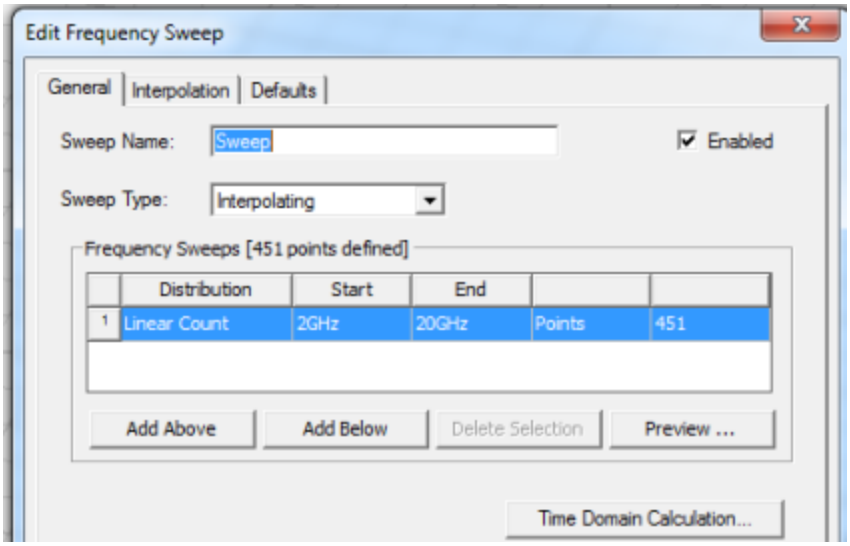
> **Note:**
>
> If you want to create a Frequency animation of field results, you must specify the frequencies here in the Multi-Frequency **Solution** setup.

Broadband

If you select Broadband, you define the frequency band by providing the low and high frequency, with a maximum delta S (that is applied to all solved frequencies), plus the maximum number of passes. The meshing frequencies include the Low and High with the remaining points determined automatically by HFSS. When the user requests DC as the low frequency, the lowest frequency solved by the adaptive meshing process is determined by an algorithm that looks at the range of frequencies you entered.



When you OK the Solution Setup for Broadband, you are automatically presented with a dialog for specifying an Interpolating sweep.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simulation and Convergence

In multiple frequencies modes, when frequency point p convergences prior to frequency point q, keep adapting at p until q and all other points achieve convergence.

The convergence data in the solution data dialog for single frequency mode is calculated in the same way as before. In multiple frequencies mode, the convergence plotted on the convergence display item corresponds to the global maximum delta S. Convergence data such as “Max. Delta S” appear in the reporter as standard solution quantity so that you can easily plot the maximum delta S for each frequency point vs. pass.

Setting the Maximum Number of Passes

The **Maximum Number of Passes** value is the maximum number of mesh refinement cycles that you would like HFSS to perform. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis will continue unless the convergence criteria are reached.

To set the maximum number of passes for an adaptive analysis:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Number of Passes**.

For driven problems and no frequency sweeps, HFSS requires at least one adaptive pass. Entering **1** will bypass adaptive analysis, generating a solution only at the solution frequency you specified.

> **Note:**
>
> The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in HFSS requesting more memory than is available or taking excessive time to compute solutions.

To shorten the solution time for solutions with a frequency sweep, you can set the **Maximum number of passes** to 0 to skip the adaptive pass phase during simulation.

Selecting 0 passes for a setup means that when you create a Report or a Field Overlay, in the Context field, the Solutions menu will not list ‘Adaptive Passes’ and ‘Last Adaptive’ for those setups. However, you can still plot the fields based on Sweep solutions if ‘Save field’ is turned on. Also, you can plot meshes using ‘Last Adaptive’ and this ‘Last Adaptive’ should be initial mesh or mesh link source mesh plus user defined mesh (if any).

If you change **Maximum number of passes** from non-zero to 0, then all plots or reports related to Last Adaptive or Adaptive pass are deleted once you click OK in the Driven Setup window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

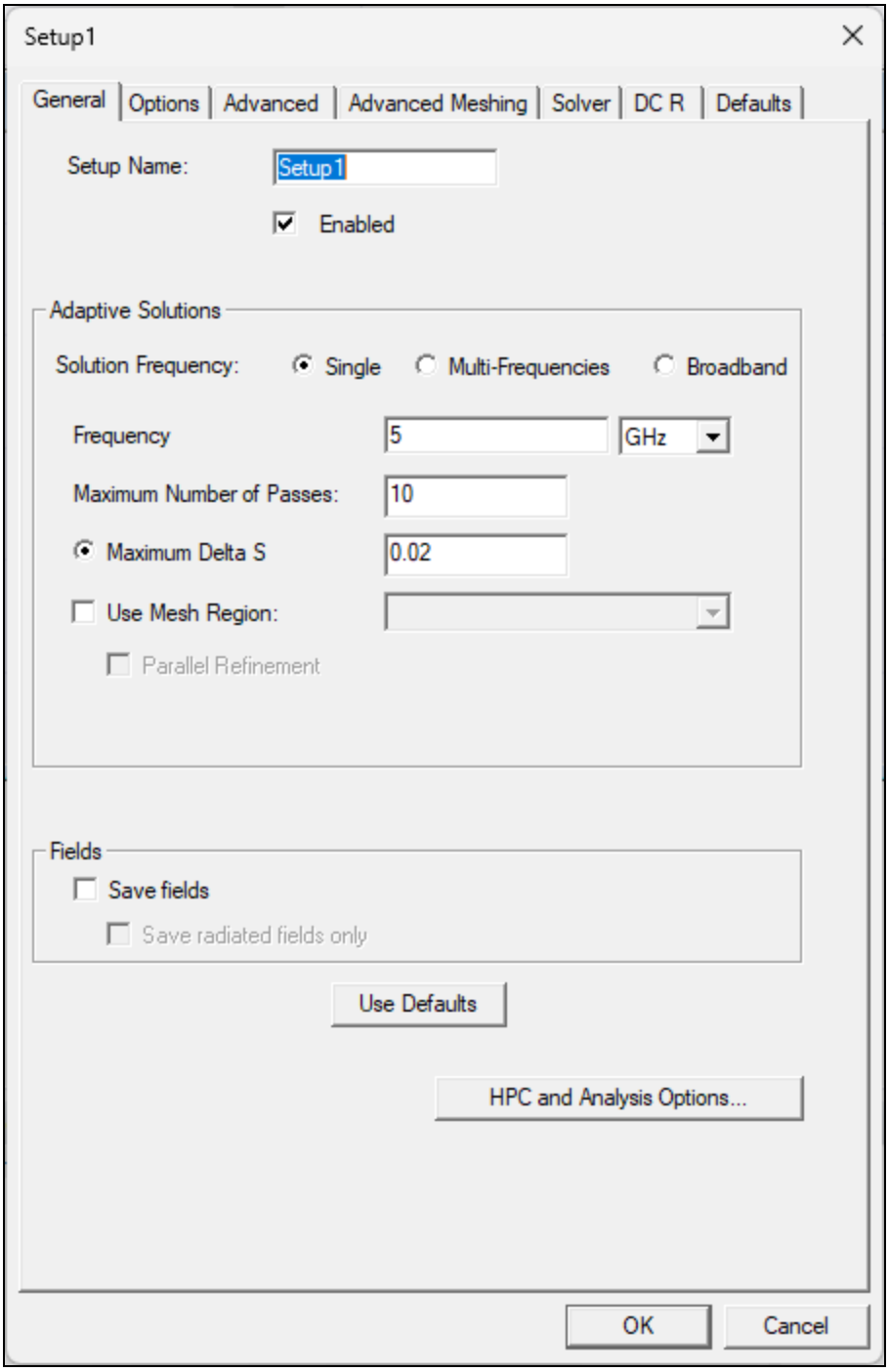
You can Undo this to recover if this was done in error. Messages in the Message window describe these actions.

If you choose 0 as the **Maximum number of passes**, the Solution Profile reports that analysis does not perform adaptive meshing.

HFSS Setup: General Tab

If applicable, select the **General** tab of the HFSS Setup window to access the following advanced HFSS solution setup settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

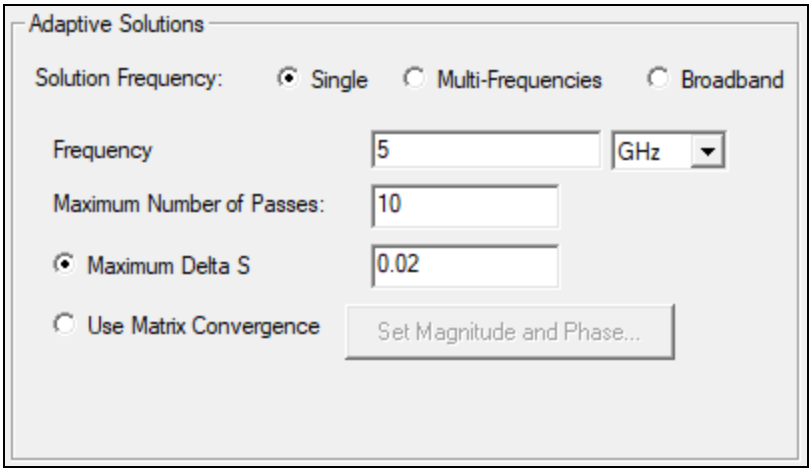


Note:

This tab is also available for the auto HFSS solution setup but with different settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Setup Name**: Name of the setup. It must be unique.
- **Enabled**: If not enabled, this setup cannot be analyzed and is not included in a general analysis of all setups.
- **Solution Frequency**: Specify the frequency and units at which to generate the solution (i.e., **Single**, **Multi-Frequencies**, or **Broadband**). If a frequency sweep is solved, an adaptive analysis is performed only at the solution frequency. Options in the **Adaptive Solutions** area change depending on the **Solution Frequency**.
    - **Single**

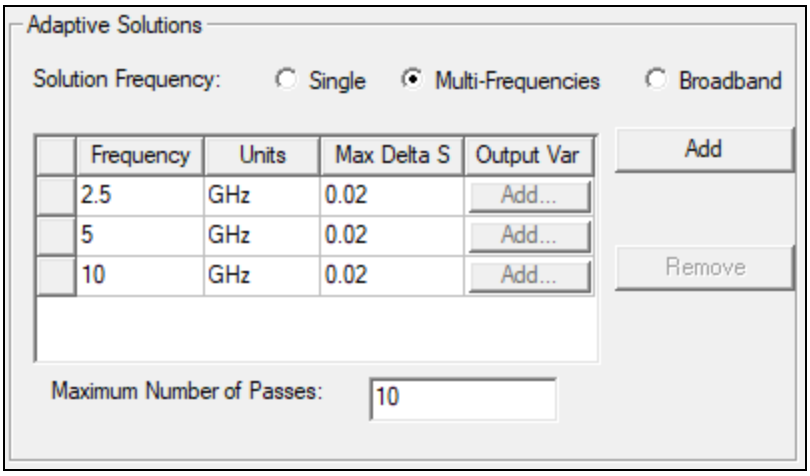


    - **Frequency**: The adaptive frequency for refinement and associated unit of measurement (i.e., from the drop-down menu, choose **Hz**, **kHz**, **MHz**, **GHz**, **THz**, **rps**, or **per_sec**).
    - **Maximum Number of Passes**:The maximum number of mesh refinement cycles that HFSS will perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis continues until the convergence criteria are reached.
    - **Maximum Delta S** or **Use Matrix Convergence**: Choose **Maximum Delta S** to affixes the magnitude of the change of the S-parameters between two consecutive adaptive passes. This is a stopping criterion for the adaptive solution. If the magnitude of the change of all S-parameters are less than this value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed. Choose **Use Matrix Convergence** to specify different stopping criteria for specific entries in the S-matrix. The adaptive analysis will continue until the magnitude and phase of the entries change by an amount less than the specified criteria from one pass to the next, or until the number of requested passes 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.

Click **Set Magnitude and Phase** to configure stopping criteria. Refer to "The Matrix Convergence Window" on page 10-19.

- **Multi-Frequencies**



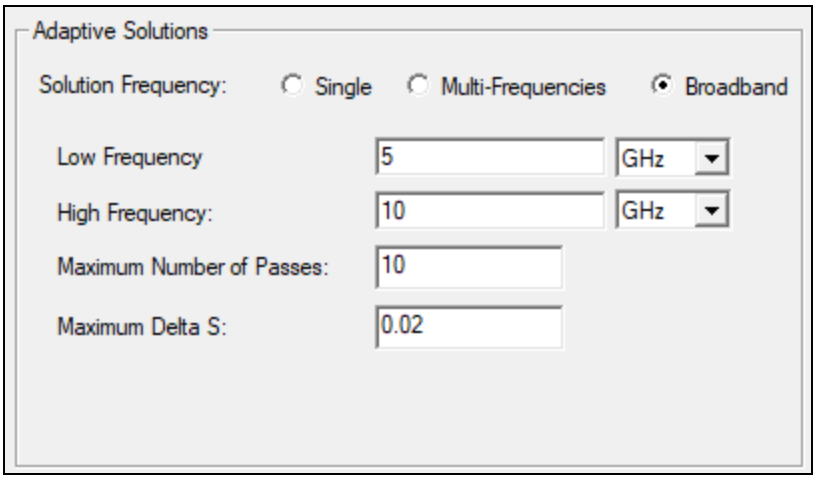
  - **Frequency**: The adaptive frequency for refinement.
  - **Units**: The associated unit of measurement (i.e., click the **Unit** field in the selected row to open a drop-down menu. Then choose **Hz**, **kHz**, **MHz**, **GHz**, **THz**, **rps**, or **per_sec**).
  - **Max Delta S**: The magnitude of the change of the S-parameters between two consecutive adaptive passes. This is a stopping criterion for the adaptive solution. If the magnitude of the change of all S-parameters are less than this value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed.
  - **Output Var.**: Click **Add** in the selected row to add output variables to the refinement criteria for that frequency. Selecting this option brings up the **Advanced Mesh Convergence** window or displays the error message 'You must have at least one output variable defined' if no output variables defined in the design. Users may receive the error 'You must have at least one output variable defined that returns a single real value (as opposed to complex, etc)'.

    - **Output Variable**: The name of the output variable.
    - **Include**: A check box for whether to include the value of this output variable in the mesh convergence calculations.
    - **Max Delta**: The maximum amount of change selected in the output variable between adaptive passes. It is a stopping criterion for the adaptive solution. If the output variable changes from one iteration to the next by an amount that is less than Max Delta, the adaptive solution stops.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Add**: Adds a new frequency to the table.
- **Remove**: Removes a selected frequency on the table.

- **Broadband**



- **Low Frequency**: The smallest adaptive frequency for refinement for a **Broadband**.
- **High Frequency**: The largest adaptive frequency for refinement for a **Broadband**.
- **Maximum Number of Passes**:The maximum number of mesh refinement cycles that HFSS will perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis continues until the convergence criteria are reached.
- **Maximum Delta S**: The magnitude of the change of the S-parameters between two consecutive adaptive passes. This is a stopping criterion for the adaptive solution. If the magnitude of the change of all S-parameters are less than this value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed.

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 **Adaptive Solutions** area includes additional parameters in IC mode:

- **Use Mesh Region**: check this box to designate specific mesh regions for enhanced simulation precision (i.e., check the box and then select a specific mesh region to refine from the drop-down menu). Refer to Creating Mesh Regions.
- **Parallel Refinement**: by default, mesh partitions are simulated sequentially for more accurate results. Check the **Parallel Refinement** box and the partitions will be simulated in parallel. Parallel Refinement is a beta feature. Beta features must be activated prior to use. Refer to Activating the Parallel Refinement Beta Feature for instructions.

- **Fields**: Click **Save fields** to post-process or plot fields. This is only available for Discrete Sweep Types. To restrict the fields saved to radiated fields, click **Save radiated fields only**.
- **Use Defaults**: The target defaults depend on the Order of Basis function selections (e.g., for Driven solutions and a First Order basis function, the default target is 0.3333. HFSS refines the mesh until most element lengths are approximately one-third wavelength.).
- **HPC and Analysis Options**: Refer to Setting HPC and Analysis Options.

When configuration is complete, click **OK** to close the **HFSS Setup** window.

The Matrix Convergence Window

Note: For designs with ports.

Choose **Use Matrix Convergence** to specify different stopping criteria for specific entries in the S-matrix. The adaptive analysis will continue until the magnitude and phase of the entries change by an amount less than the specified criteria from one pass to the next, or until the number of requested passes is completed. Setup Matrix Convergence by completing these steps.

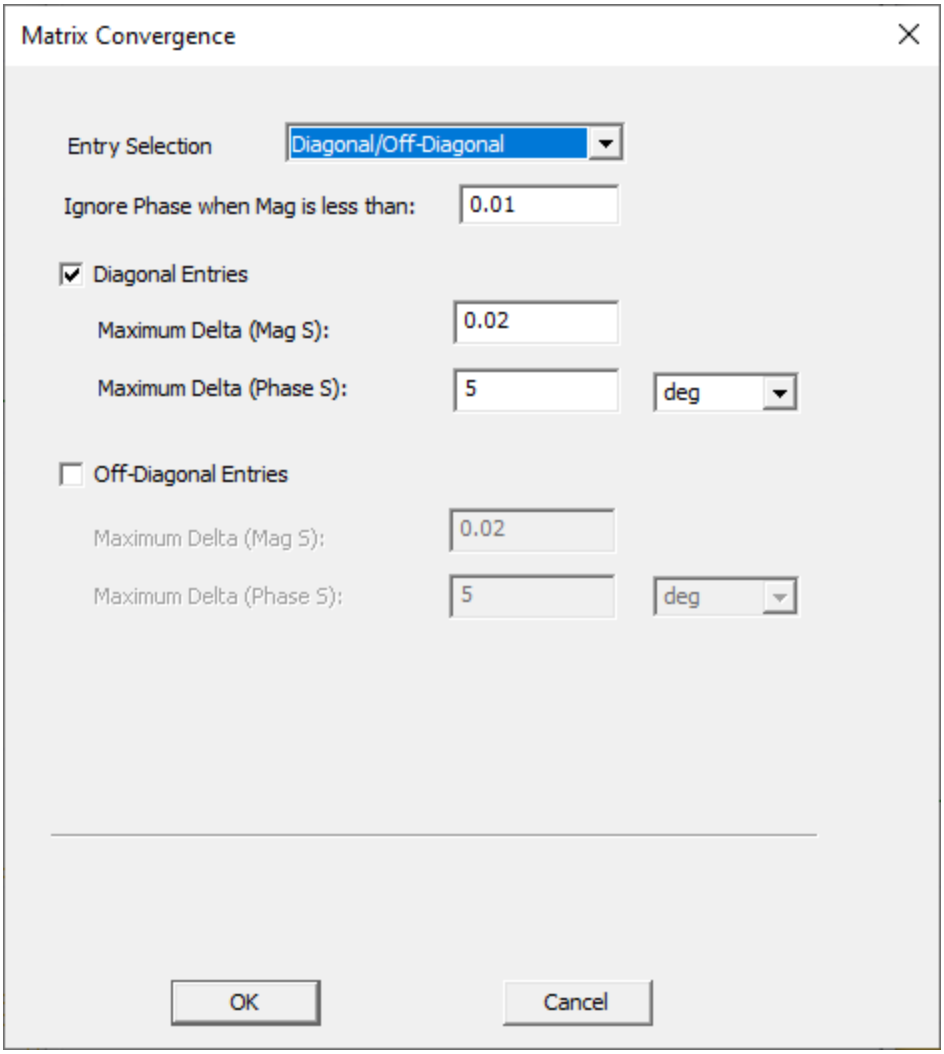
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When the Mag S becomes small (near to zero) its phase becomes indefinite and insignificant due to mathematical issues. In HFSS 10 and 11 there was a 0.05 magnitude threshold that caused that Phase Margin to be discarded. However, some users may want to continue running additional passes to stabilize phase margins even when the S-parameter magnitude is below this threshold. For this reason, the magnitude threshold has been removed. This calls for caution: to see a good phase convergence for S_{12}, which is near to 1. If a small delta_phi with small delta_s is set, after a couple of passes, S_{12} will converge, but S_{11} never, since S_{11} ~0 and its phase changes with the mesh noise because the phase is indefinite. In other words, under these conditions the adaptive process never stops, so abort it.

1. From the **HFSS Setup** window, choose **Use Matrix Convergence**. Then click **Set Magnitude and Phase** to open the **Matrix Convergence** window.

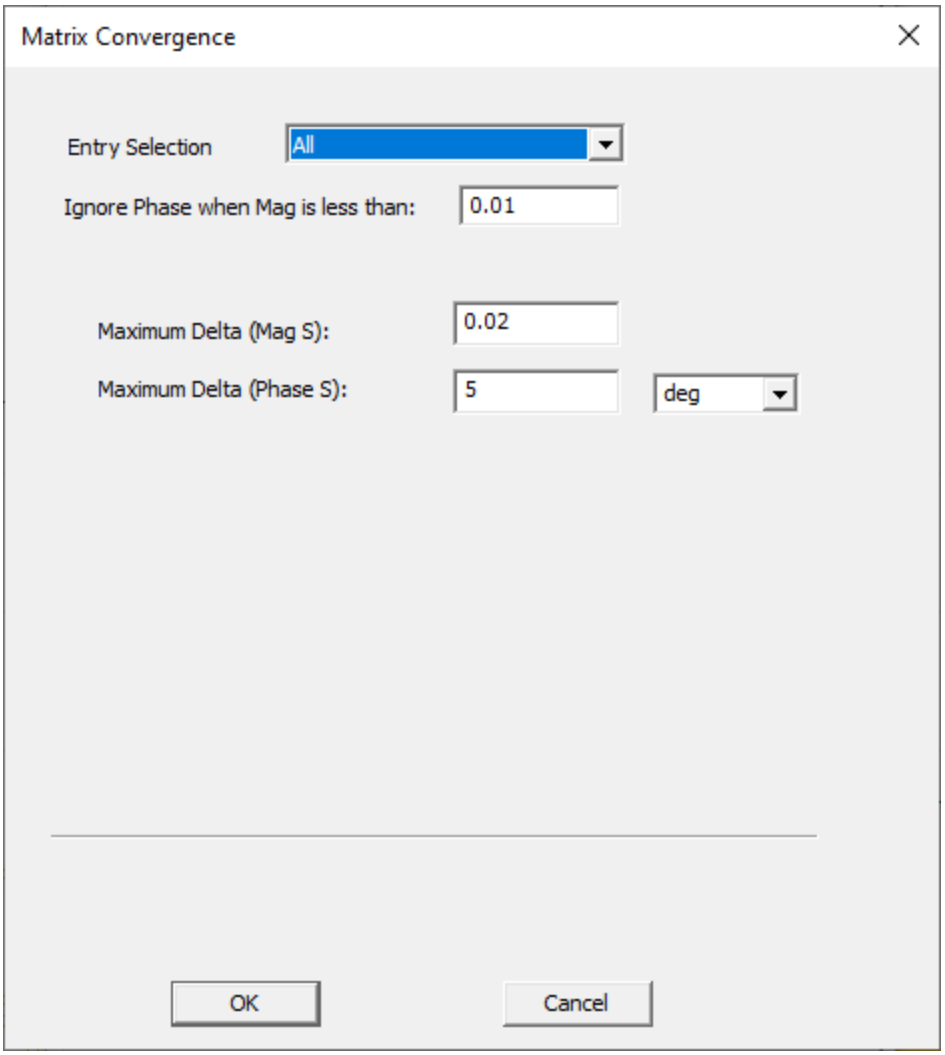
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Options in the **Matrix Convergence** window change depending on the **Entry Selection**. Choose one of the following on the Entry Selection 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.

- **All**: Sets all the matrix entries at once. (The default).

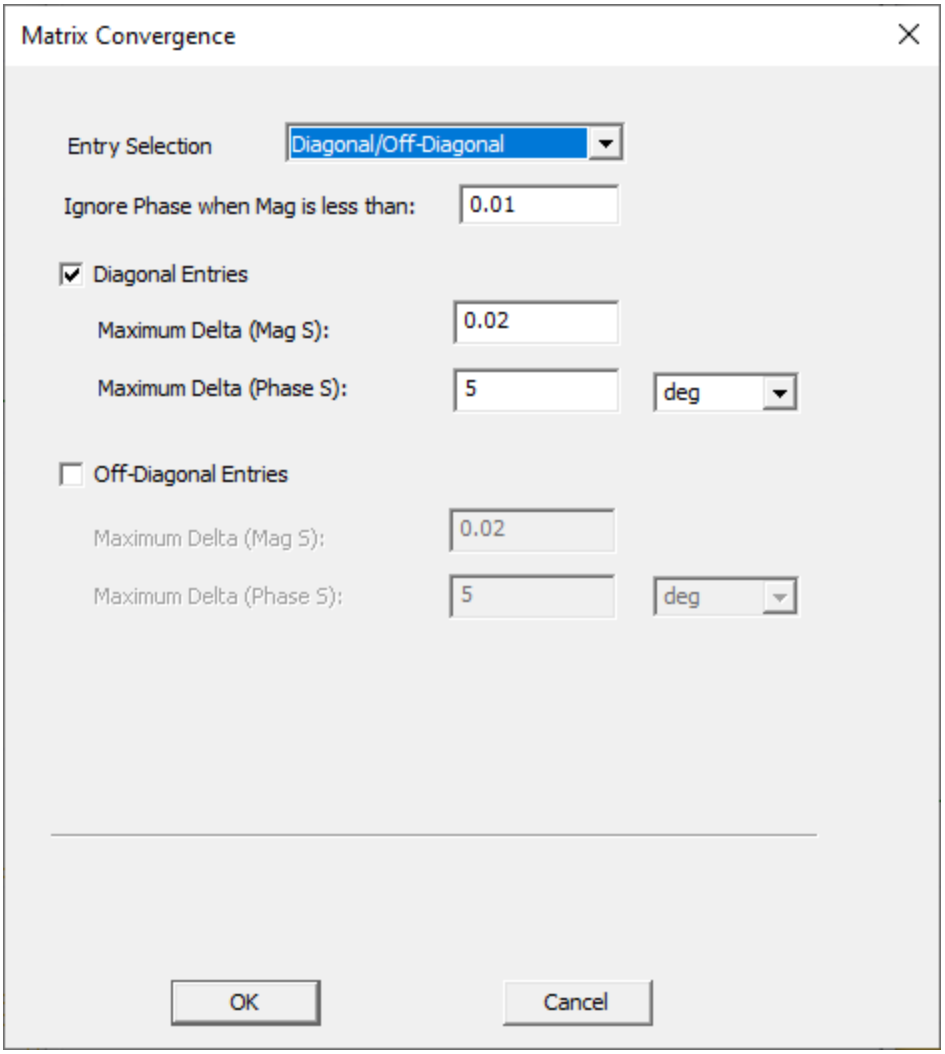


  a. If applicable, enter a value in the **Ignore Phase when Mag is less than:** field.

  b. Enter the convergence criteria for the **Maximum Delta (Mag S)** and the **Maximum Delta (Phase S)** in the selected fields, and choose a unit of measurement from the drop-down menu (i.e., **deg**, **degmin**, **degsec**, **rag**).

- **Diagonal/Off-Diagonal**: Sets all the diagonal matrix entries at once, all off-diagonal matrix entries at once, or both diagonal and off diagonal entries at once.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



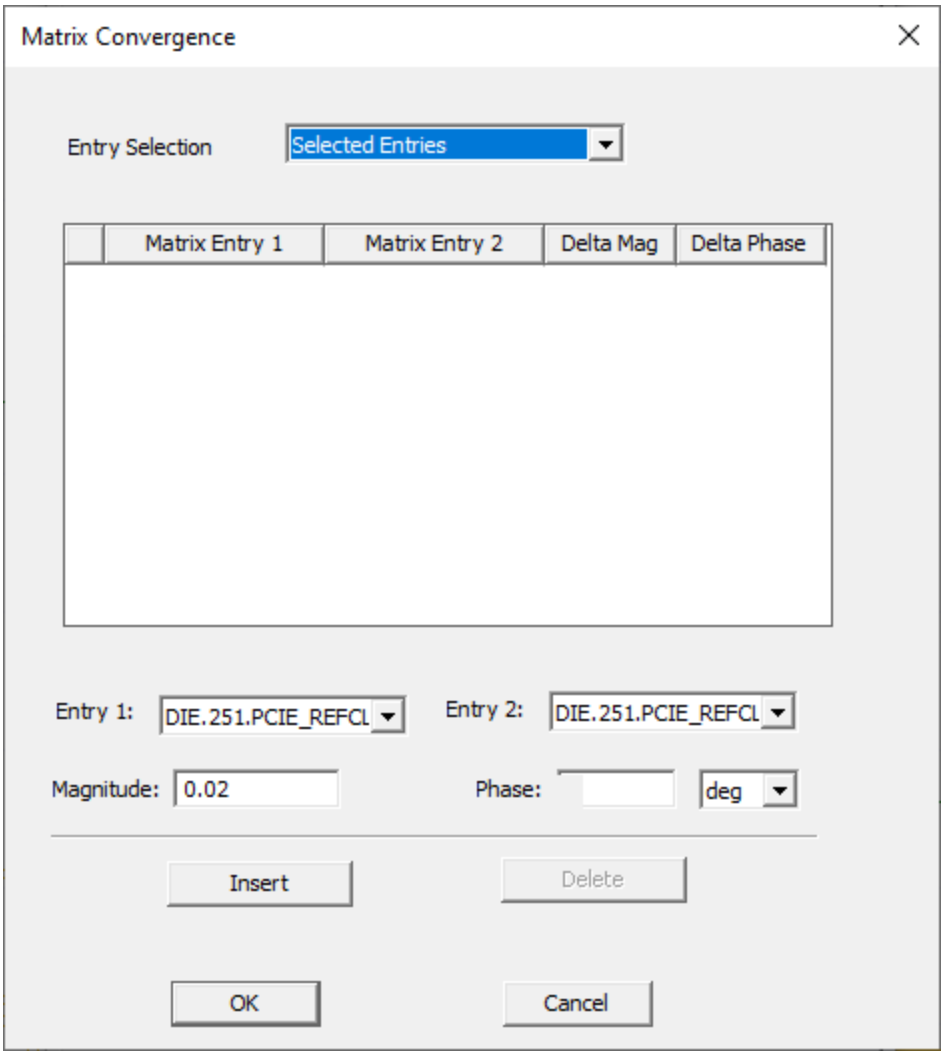
a. If applicable, enter a value in the **Ignore Phase when Mag is less than:** field.

b. Check **Diagonal Entries**, **Off-Diagonal Entries**, or both, to enable the convergence criteria field(s).

c. Enter the convergence criteria for the **Maximum Delta (Mag S)** and the **Maximum Delta (Phase S)** in the appropriate fields, and choose a unit of measurement from the drop-down menu (i.e., **deg**, **degmin**, **degsec**, **rag**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Selected Entries**: Sets individual matrix entries that users select.



Do the following to select the appropriate ports and mode (or terminal) pairs:

a. Select **Entry 1:** and **Entry 2:** from the drop-down menus.

b. From the **Magnitude:** field, enter the maximum change in magnitude from **Entry 1** (first pass) to **Entry 2** (second pass).

c. From the **Phase:** field, enter the maximum change in phase from **Entry 1** (first pass) to **Entry 2** (second pass), and choose a unit of measurement from the drop-down menu (i.e., **deg**, **degmin**, **degsec**, **rag**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Click **Insert** and the selected perimeters will appear in the table, dispersed into fields in the **Matrix Entry 1**, **Matrix Entry 2**, **Delta Mag**, and **Delta Phase** columns.

e. Repeat steps a-d, as appropriate.

f. Select a row in the table to enable the **Delete** button. Click **Delete** to remove the row, as appropriate.

g. Clicking in the **Delta Mag** and **Delta Phase** columns enables editing in the selected field.

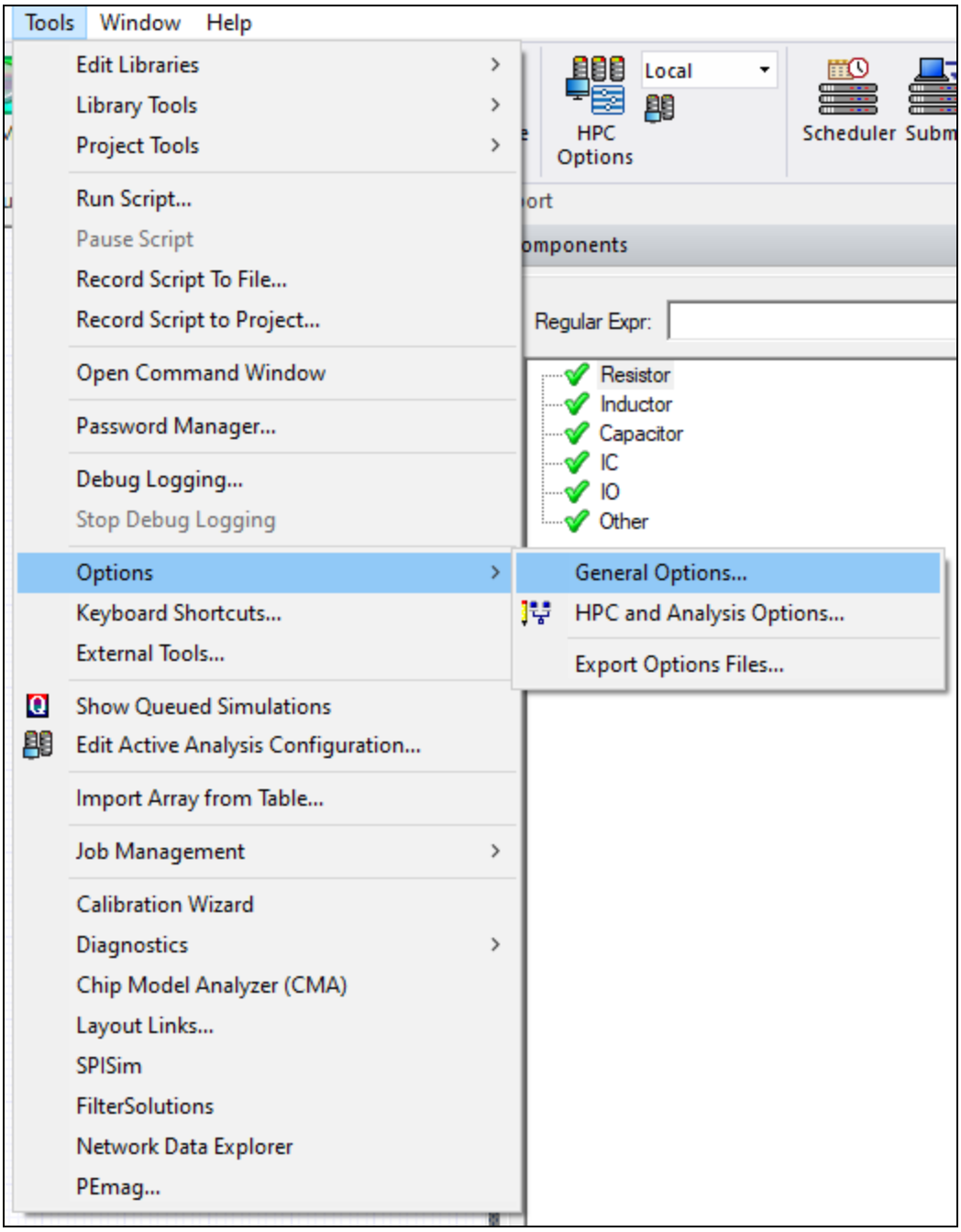
When configuration is complete, click **OK** to close the **Matrix Convergence** window.

Activating the Parallel Refinement Beta Feature

Follow these steps to enable the parallel refinement beta feature.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

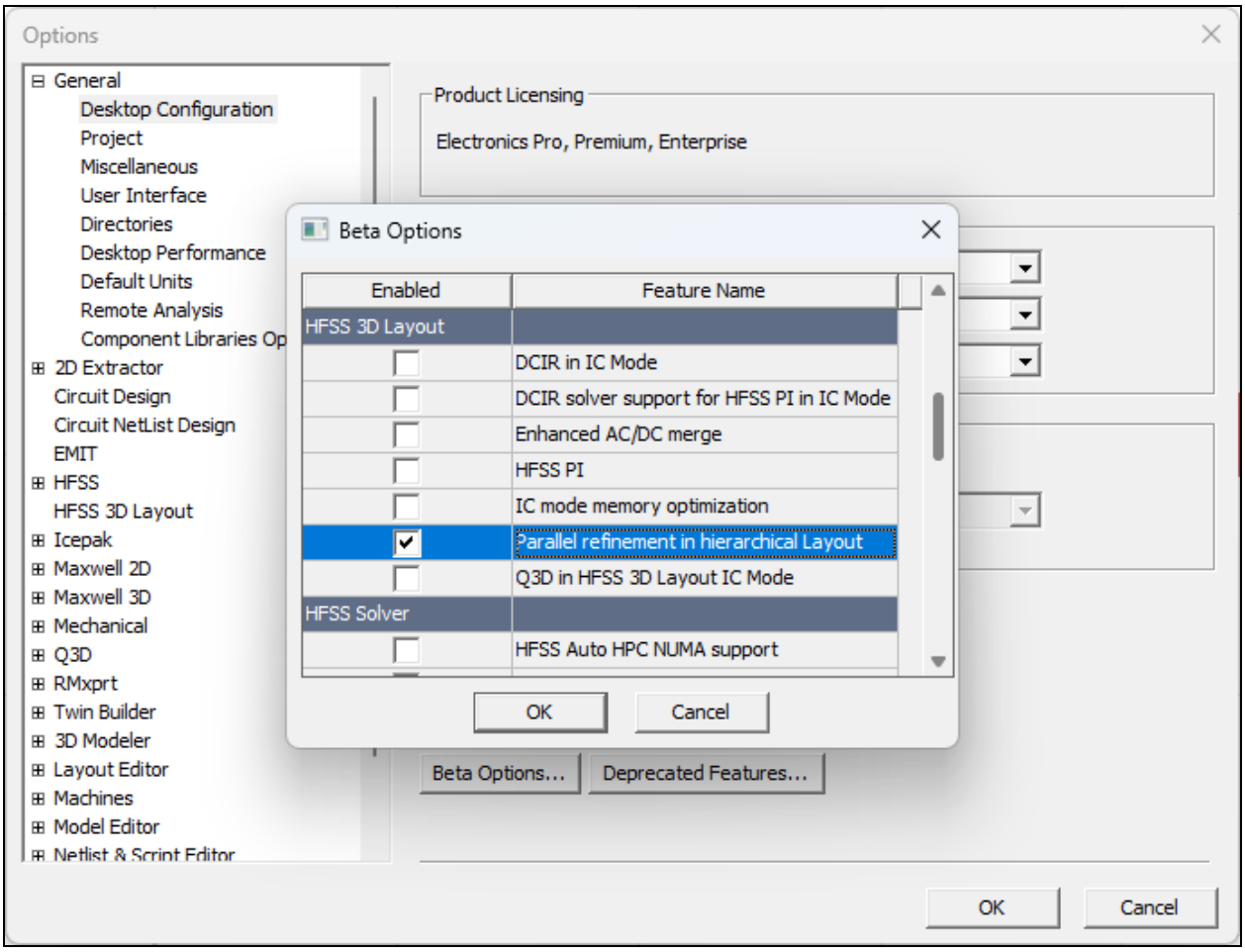
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window



2. Expand **General** > **Desktop Configuration** and click **Beta Options** to open the **Beta 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.

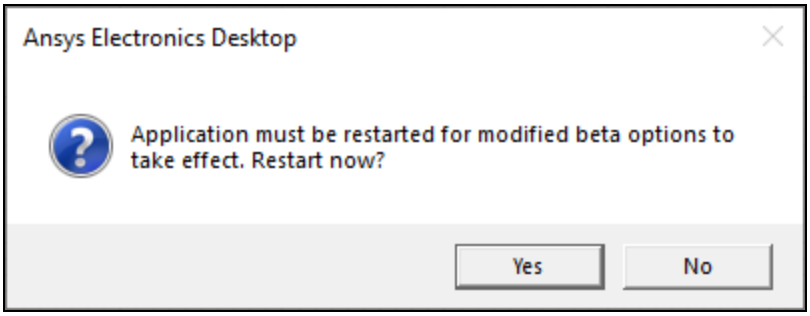
3. Scroll down to **Parallel refinement in hierarchical Layout** and check the adjacent box



4. Click **OK** to close the **Beta Options** window.

5. Click **OK** to close the **Options** window and open a prompt to restart the software. Users may be asked to save their work.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



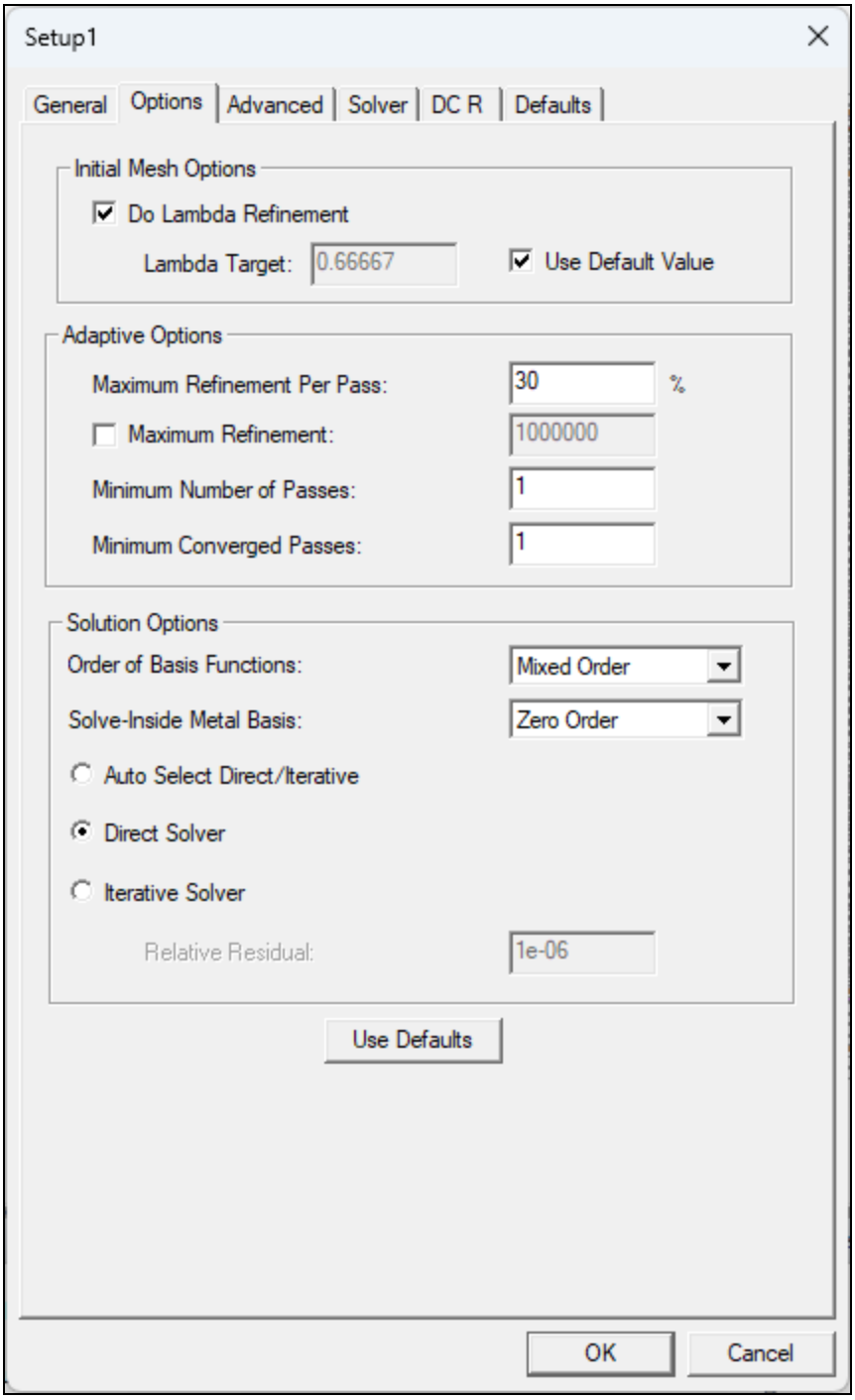
6. Click **Yes** to restart immediately.

After restarting the software, the parallel refinement option is available from the locations described in the previous section.

HFSS Setup: Options Tab

Select the **Options** tab of the **HFSS Setup** window to access the following settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Initial Mesh Options: options that define the initial mesh before adaption.

- **Do Lambda Refinement**: Lambda refinement is the process of refining the initial mesh based on the material-dependent wavelength.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Lambda Target**: The fraction of the wavelength that determines the refined tetrahedra edge length (e.g., if the value is 0.1 then a edge length of 0.1 is used as the refinement target).
- **Use Default Value**: The target defaults depend on the Order of Basis function selections (e.g., for Driven solutions and a First Order basis function, the default target is 0.3333; HFSS refines the mesh until most element lengths are approximately one-third wavelength.).

Adaptive Options:

- **Maximum Refinement Per Pass**: determines how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error is refined. The value is a percentage.
- **Maximum Refinement**: The maximum number of tetrahedra that can be added during an adaptive pass. By default, this unchecked, to there is no maximum. If you enable the Maximum Refinement, the initial value is 1000000.
- **Minimum Number of Passes**: The maximum number of mesh refinement cycles that you want HFSS to perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis continues unless the convergence criteria are reached.
- **Minimum Converged Passes**: An adaptive analysis does not stop unless the minimum number of converged passes has been completed. The convergence criteria must be met for at least this number of passes before the adaptive analysis stops.

Solution Options:

- **Order of Basis Functions**: The order of the basis functions HFSS uses to interpolate field values from nodal values. The **Zero Order** option is useful when a model requires a mesh that produces more than 100,000 tetrahedra, but the model size is small compared to wavelength. The higher order options (i.e., **First** and **Second Order**) solve progressively more unknowns for each tetrahedra. **Mixed Order** uses higher order where more accuracy is required, and lower order where fields are weaker.
- **Solver-Inside Metal Basis (IC Mode Only)**: determines the order of basis functions used inside metallic objects. The **Zero Order** option uses lower-order interpolation, reducing computation time and memory usage in most designs. The **First Order** option uses a higher-order basis for increased accuracy inside metal regions. Recommended if currents inside metal regions would significantly impact results.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> If **Zero Order** is selected from the **Order of Basis Functions** drop-down menu, only **Zero Order** can be selected from the **Solver-Inside Metal Basis** drop-down menu.

- **Auto Select Direct/Iterative**: The auto solver automatically chooses whether to use the direct (i.e., default) or iterative solver **for each pass**. The direct solver provides a multi-frontal (MF) approach and is invoked by default if the matrix is not well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also reduce solution time for large simulations.
- **Enable Direct Solver** (i.e., default): The direct solver is a multi-frontal solver ideal for smaller simulations.
- **Enable Iterative Solver**: The iterative solver provides an alternative to the multi-frontal solver when a matrix is well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also provide a savings in the solution time for large simulations. When this option is enabled, HFSS automatically invokes the iterative solver when it decides the matrix is conditioned well enough to take advantage of the iterative approach. HFSS uses the multi-frontal solver if the matrix does not meet this requirement.
- **Relative Residual**: The residual measures the convergence of the iterative solver. The default value is 1E-4.
- **Enhanced low frequency accuracy**: When enabled, the solver is tuned to reliably solve low frequencies for designs that only contain lumped ports as sources or a combination of lumped and circuit ports. For designs with only lumped ports and/or circuit ports, interpolating sweeps are tuned to solve more low frequency points in order to accurately represent very low frequency results.

> **Important:**
>
> **Enhanced low frequency accuracy** has no affect on **Discrete** sweeps.

> **Note:**
>
> **Enhanced low frequency accuracy** does not support wave ports at this time.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> **Enhanced low frequency accuracy** requires more RAM in order to solve the entire solution vector for interpolation sweeps and for discrete sweeps not saving fields. For more information, see Horizontal Lumped Ports.

Enable Auto Select Direct/Iterative

In order to reduce the need for user intervention, speed up the solution process, and minimize solver failure, HFSS automatically chooses whether to use the direct (i.e., default) or iterative solver **for each pass**. Depending on a wide range of parameters (e.g., geometry, mesh size and quality, basis order, the number of excitations, et cetera), the performance of the direct and iterative solvers can vary significantly. The direct solver provides a multi-frontal (MF) approach and is invoked by default if the matrix is not well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also reduce solution time for large simulations. Enabling **Auto Select Direct/Iterative** could therefore potentially improve the speed of the simulation while maintaining accuracy.

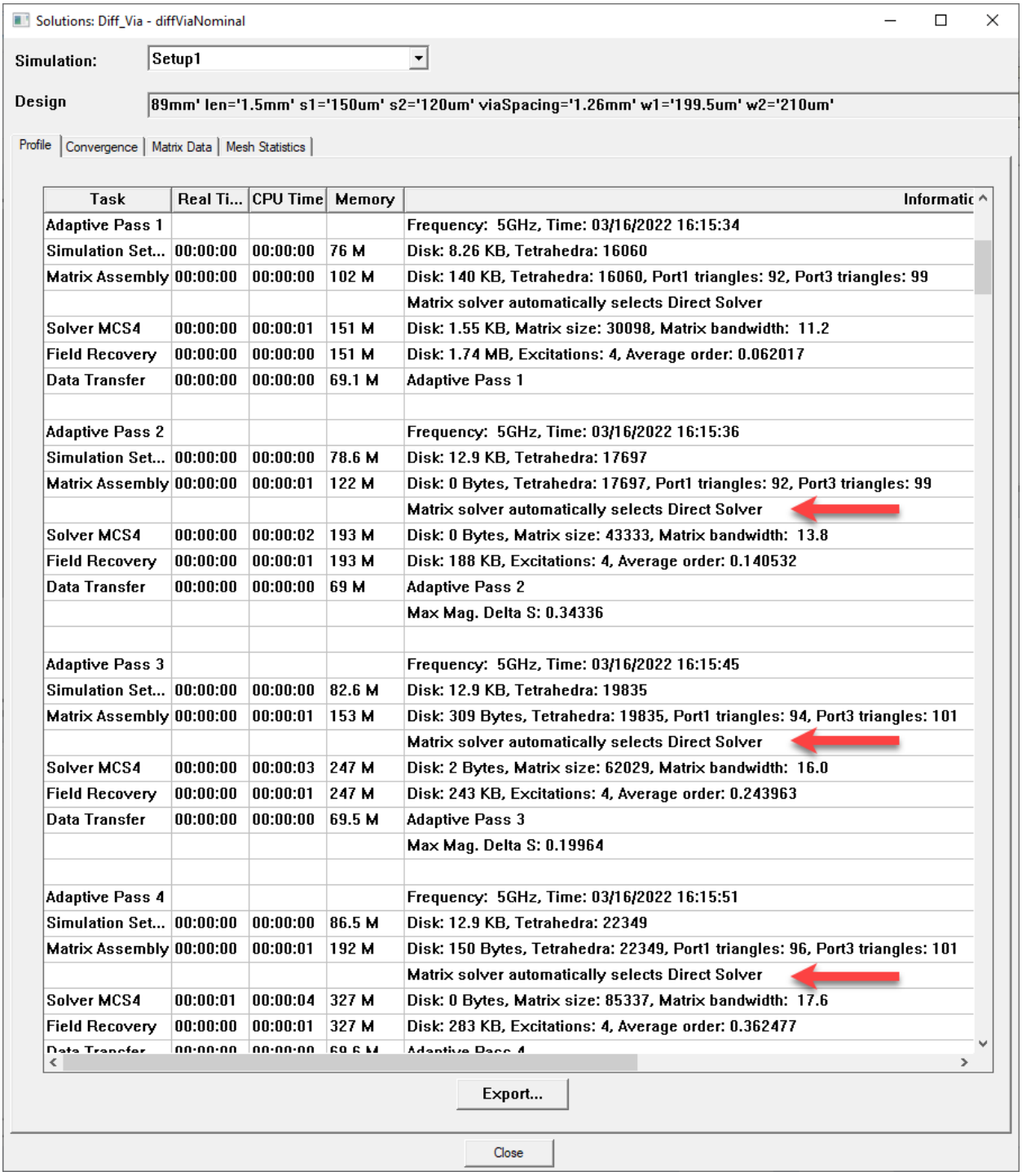
Enable the Auto Select Direct/Iterative

From the **Solution Setup** window, select the **Options** tab. Then select **Auto Select Direct/Iterative** on the **Solution Options** area.

> **Note:**
>
> The Iterative Solver is not available for zero order basis solutions.

During and after analysis, the solver selected for each pass and sweep is viewable on the **Solutions** window > **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 Ti...	CPU Time	Memory	Informatic
Adaptive Pass 1				Frequency: 5GHz, Time: 03/16/2022 16:15:34
Simulation Set...	00:00:00	00:00:00	76 M	Disk: 8.26 KB, Tetrahedra: 16060
Matrix Assembly	00:00:00	00:00:00	102 M	Disk: 140 KB, Tetrahedra: 16060, Port1 triangles: 92, Port3 triangles: 99
				Matrix solver automatically selects Direct Solver
Solver MCS4	00:00:00	00:00:01	151 M	Disk: 1.55 KB, Matrix size: 30098, Matrix bandwidth: 11.2
Field Recovery	00:00:00	00:00:00	151 M	Disk: 1.74 MB, Excitations: 4, Average order: 0.062017
Data Transfer	00:00:00	00:00:00	69.1 M	Adaptive Pass 1
Adaptive Pass 2				Frequency: 5GHz, Time: 03/16/2022 16:15:36
Simulation Set...	00:00:00	00:00:00	78.6 M	Disk: 12.9 KB, Tetrahedra: 17697
Matrix Assembly	00:00:00	00:00:01	122 M	Disk: 0 Bytes, Tetrahedra: 17697, Port1 triangles: 92, Port3 triangles: 99
				Matrix solver automatically selects Direct Solver
Solver MCS4	00:00:00	00:00:02	193 M	Disk: 0 Bytes, Matrix size: 43333, Matrix bandwidth: 13.8
Field Recovery	00:00:00	00:00:01	193 M	Disk: 188 KB, Excitations: 4, Average order: 0.140532
Data Transfer	00:00:00	00:00:00	69 M	Adaptive Pass 2
				Max Mag. Delta S: 0.34336
Adaptive Pass 3				Frequency: 5GHz, Time: 03/16/2022 16:15:45
Simulation Set...	00:00:00	00:00:00	82.6 M	Disk: 12.9 KB, Tetrahedra: 19835
Matrix Assembly	00:00:00	00:00:01	153 M	Disk: 309 Bytes, Tetrahedra: 19835, Port1 triangles: 94, Port3 triangles: 101
				Matrix solver automatically selects Direct Solver
Solver MCS4	00:00:00	00:00:03	247 M	Disk: 2 Bytes, Matrix size: 62029, Matrix bandwidth: 16.0
Field Recovery	00:00:00	00:00:01	247 M	Disk: 243 KB, Excitations: 4, Average order: 0.243963
Data Transfer	00:00:00	00:00:00	69.5 M	Adaptive Pass 3
				Max Mag. Delta S: 0.19964
Adaptive Pass 4				Frequency: 5GHz, Time: 03/16/2022 16:15:51
Simulation Set...	00:00:00	00:00:00	86.5 M	Disk: 12.9 KB, Tetrahedra: 22349
Matrix Assembly	00:00:00	00:00:01	192 M	Disk: 150 Bytes, Tetrahedra: 22349, Port1 triangles: 96, Port3 triangles: 101
				Matrix solver automatically selects Direct Solver
Solver MCS4	00:00:01	00:00:04	327 M	Disk: 0 Bytes, Matrix size: 85337, Matrix bandwidth: 17.6
Field Recovery	00:00:00	00:00:01	327 M	Disk: 283 KB, Excitations: 4, Average order: 0.362477
Data Transfer	00:00:00	00:00:00	69.6 M	Adaptive Pass 4

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable Direct Solver

The direct solver provides a multi-frontal (MF) approach when a matrix is not well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also provide a savings in the solution time for large simulations.

When you select the **Enable Direct Solver** option, HFSS automatically invokes the multi-frontal solver. If you select **Iterative Solver** and HFSS decides that the matrix is not conditioned well enough to take advantage of the iterative approach HFSS still uses the multi-frontal solver.

Enable the Direct Solver

From the **Solution Setup** window, select the **Options** tab. Then select **Direct Solver** from the *Solution Options* group box.

> **Note:** The Iterative Solver is not available for zero order basis solutions.

Enable Iterative Solver

The iterative solver provides an alternative to the multi-frontal solver when a matrix is well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also provide a savings in the solution time for large simulations.

When you select **Enable Iterative Solver**, HFSS automatically invokes the iterative solver when it decides that the matrix is conditioned well enough to take advantage of the iterative approach. HFSS uses the multi-frontal solver if the matrix does not meet this requirement.

For more detail, see the technical notes for Iterative Matrix Solver.

Enable the Iterative Solver

From the **Solution Setup** window, select the **Options** tab. Then select **Iterative Solver** from the *Solution Options* group box.

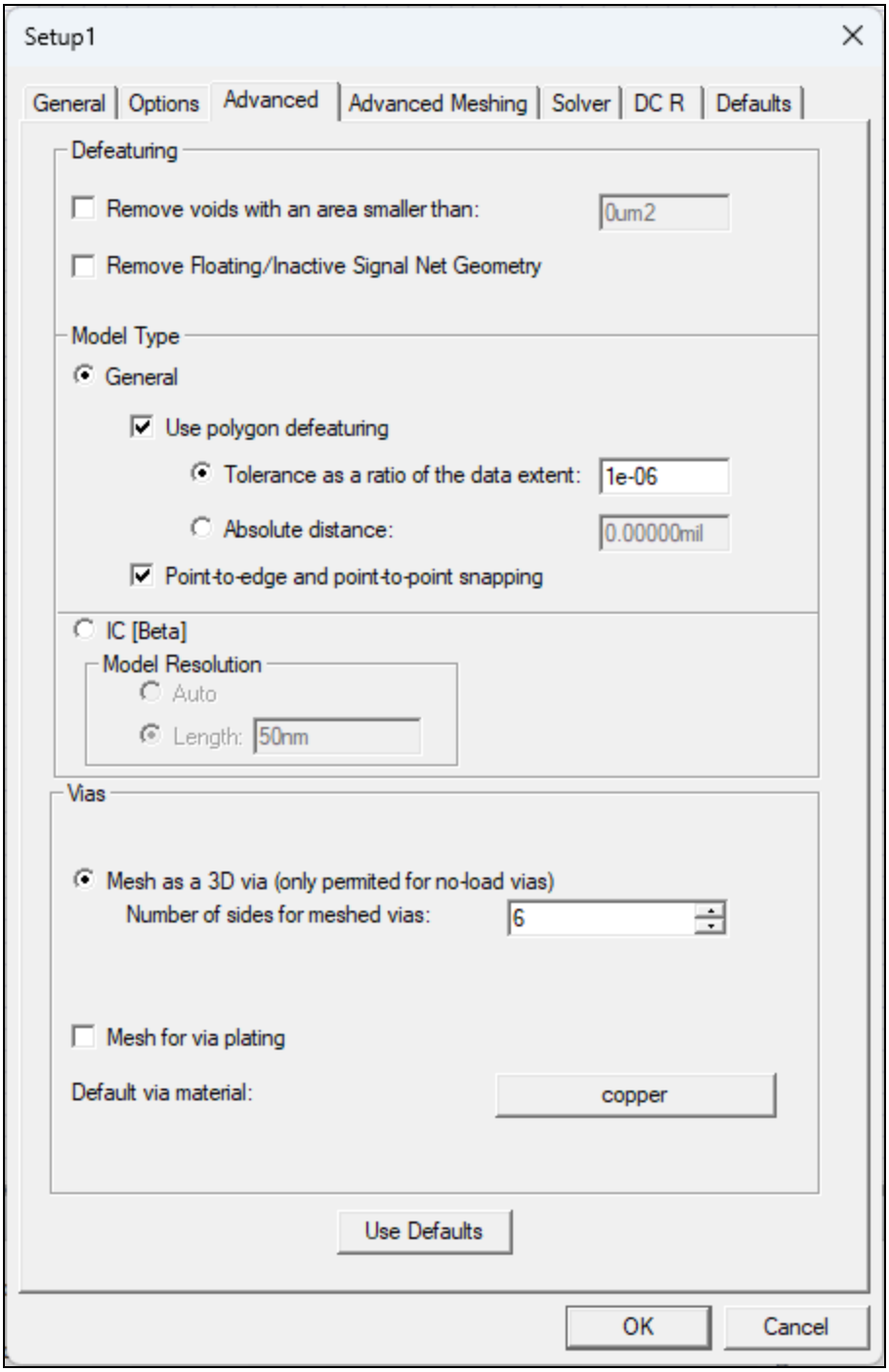
This enables the **Relative Residual** check box. The residual measures the convergence of the iterative solver. Check the box, if desired, and enter a **Relative Residual** value in the adjacent text field (default value is 1E-6).

> **Note:** The Iterative Solver is not available for zero order basis solutions.

HFSS Setup: Advanced Tab

Select the **Advanced** tab of the HFSS Setup window to access the following settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defeaturing

- **Remove voids with an area smaller than** <value>: simplifies meshing by removing small voids.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Remove Floating/Inactive Signal Net Geometry**: removes floating and inactive signal net geometry.

Model Resolution (IC Mode Only)

- **Auto**: automatically sets the model resolution.
- **Length**: allows you to manually set the model resolution.

> **Note:**
>
> The following options are only available in General mode.

Model Type: offers two types of models.

- **General**: offers standard mesh processing.
  - **Use polygon defeaturing**: removes very close points, points that do not contribute to the geometry of an edge (e.g., collinear), and very thin intrusions. When used with “unioning”, it can be helpful in healing geometry that is not snapped together.
    - **Tolerance as a ratio of the data extent**: tolerance value specified as a ratio of the overall data extent. This value must be very small. Useful for rigid-flex bends. Refer to Defining Rigid-Flex Bends.
    - **Absolute distance**: tolerance specified as an absolute value (e.g., 0.0001mm).

    > **Note:**
    >
    > From the **Vias** area, vias may be modeled as a simple 'wirebond' or as a 3D ribbon or solid. The values specified in the analysis setup only apply to vias that do not have specific property overrides; all properties explicitly specified for a via take precedence over these values.

  - **Point -to-edge and point-to-point snapping**: snaps points to edges and points to points if they are within a minimum dimensional tolerance. This tolerance is based on geometry extents.
- **IC [Beta]**: optimizes geometry processing to simulate small, on-chip structures.
  - **Auto**: automatically sets the model resolution based on the design's geometry. Recommended for most designs.
  - **Length**: allows the user to manually set a resolution threshold. Features smaller than the entered value may be merged with nearby geometry, snapped, or removed entirely, especially if they don’t affect the overall structure or function of the design. Recommended if the design includes fine or critical details.

Vias

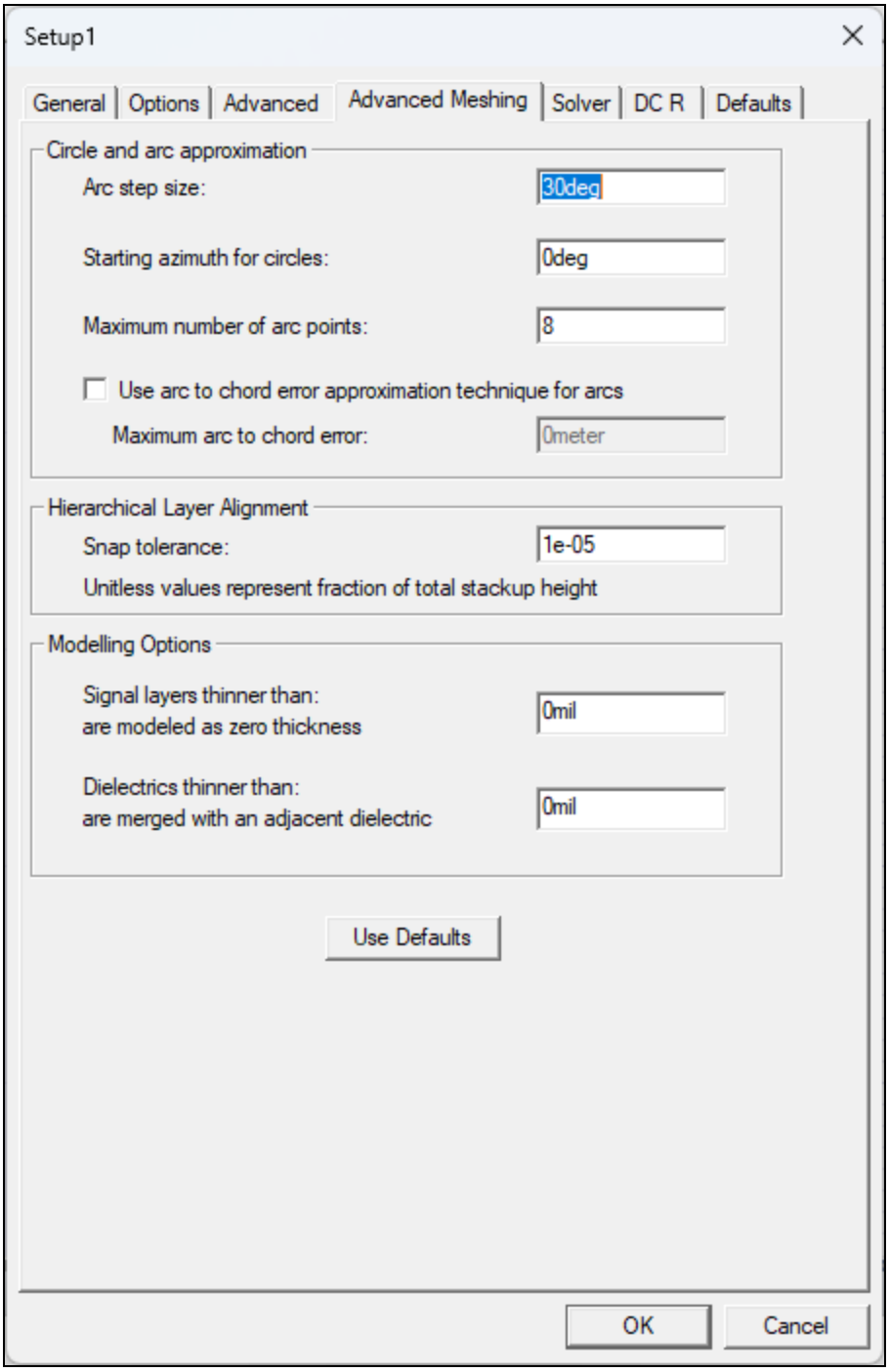
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh as a 3D via**: vias are modeled using rectangles to form a 3D ribbon or solid.
  - **Number of sides**: number of sides to use when creating the 3D representation. If less than 3, the via is modeled as a flat ribbon.
  - **Mesh for via plating**: check the box to create a mesh around the plating percent specified in the **Edit Padstack Definition** window, resulting in a more accurate DC thickness value. A via that is not fully plated includes a void filled with the material defined in the surrounding air box (e.g., air). Zero plating is not recognized and is treated as fully plated.
- **Default via material**: when a material has not been specified in the padstack definition and there is no override in the properties for a via, then this is the material associated with the via mesh. It is initially set to copper.

HFSS Setup: Advanced Meshing Tab

Select the **Advanced Meshing** tab of the HFSS Setup window to access the following settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Circle and arc approximation**: circles and arcs must be replaced with straight edge approximations before meshing. There is no attempt to match areas. Points are introduced on the original arc at the locations specified by the following parameters.

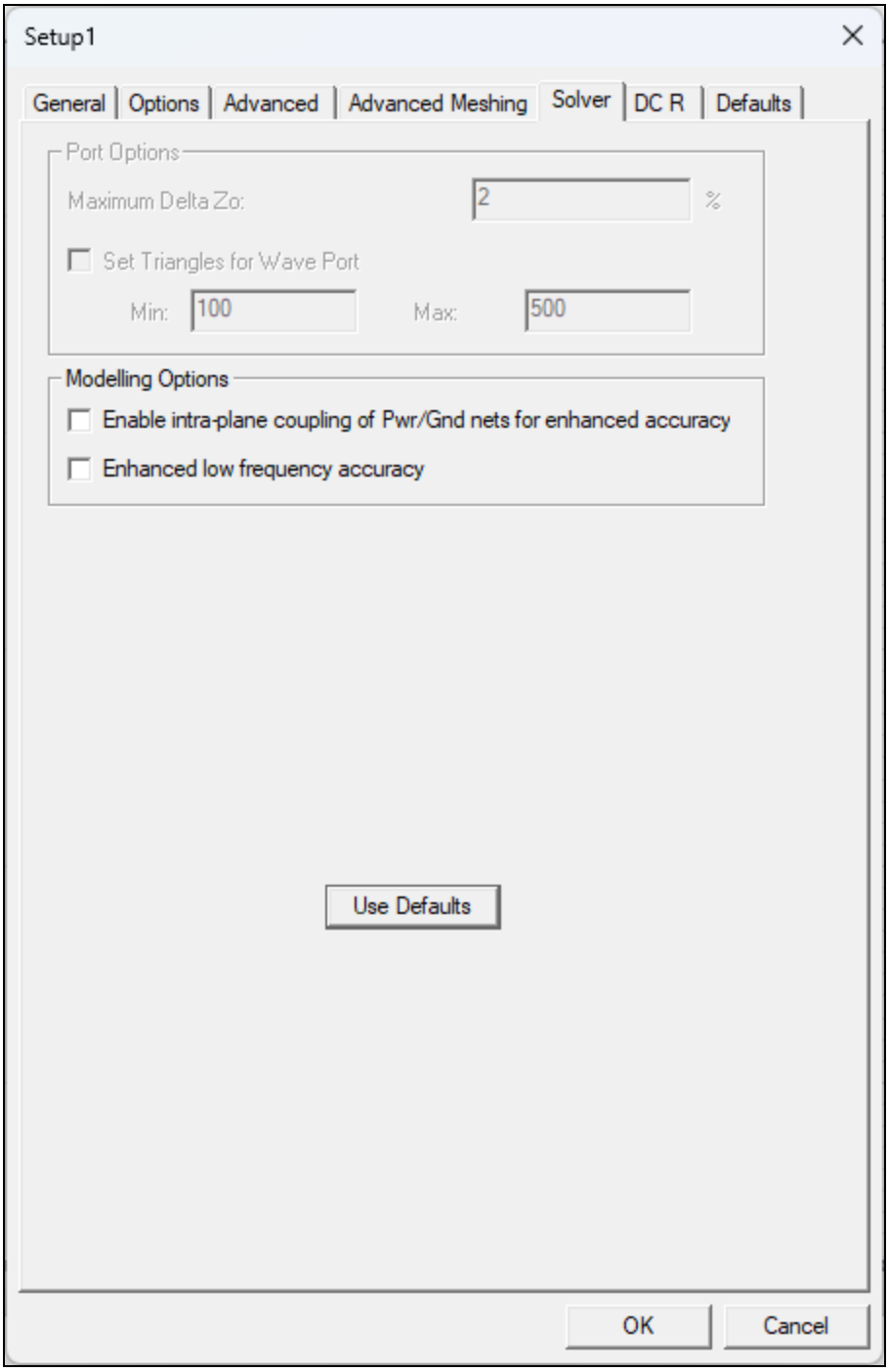
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Arc step size**: the angular interval below which mesh points are added.
  - **Starting azimuth for circles**: specifies the location of the first mesh point or circles.
  - **Maximum number of arc points**: the maximum number of mesh points on a given arc segment.
  - **Use arc to chord error approximation technique for arc**: the resulting straight edge approximation deviate from the original arc by more than the specified error.
    - **Maximum arc to chord error**: maximum distance permissible between a straight edge approximation (the chord) and the original arc.
- **Hierarchical Layer Alignment**: aligns layers across hierarchical boundaries.
  - **Snap tolerance**: When layers across hierarchical boundaries are within this unitless tolerance, they are aligned to the same elevation. This value represents a fraction of total stackup height.
  - **Signal layers thinner than <*value*> are modeled as zero thickness**: enter a value. Negative signal layers in the design which are thinner than the value are modeled as being infinitely thin in order to reduce simulation time.
  - **Dielectrics thinner than <*value*> are merged with an adjacent**: enter a value. Dielectric layers in the design which are thinner than the value are merged into an adjacent dielectric. Material properties are accumulated using a weighted average.

HFSS Setup: Solver Tab

Select the **Solver** tab of the HFSS Setup window to access the following settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Port Options (General Mode Only)

- **Maximum Delta Zo**: Change to Zo specified as a target percentage. The default is 2%.
- **Set Triangles for Wave Port**: Controls the number of triangles used for a wave port. For designs with lumped ports, this option is not active. Higher numbers of triangles do not

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

benefit a solution setup in this case.

- **Min**: Minimum number of triangles on the wave port face.
- **Max**: Maximum number of triangles on the wave port face.

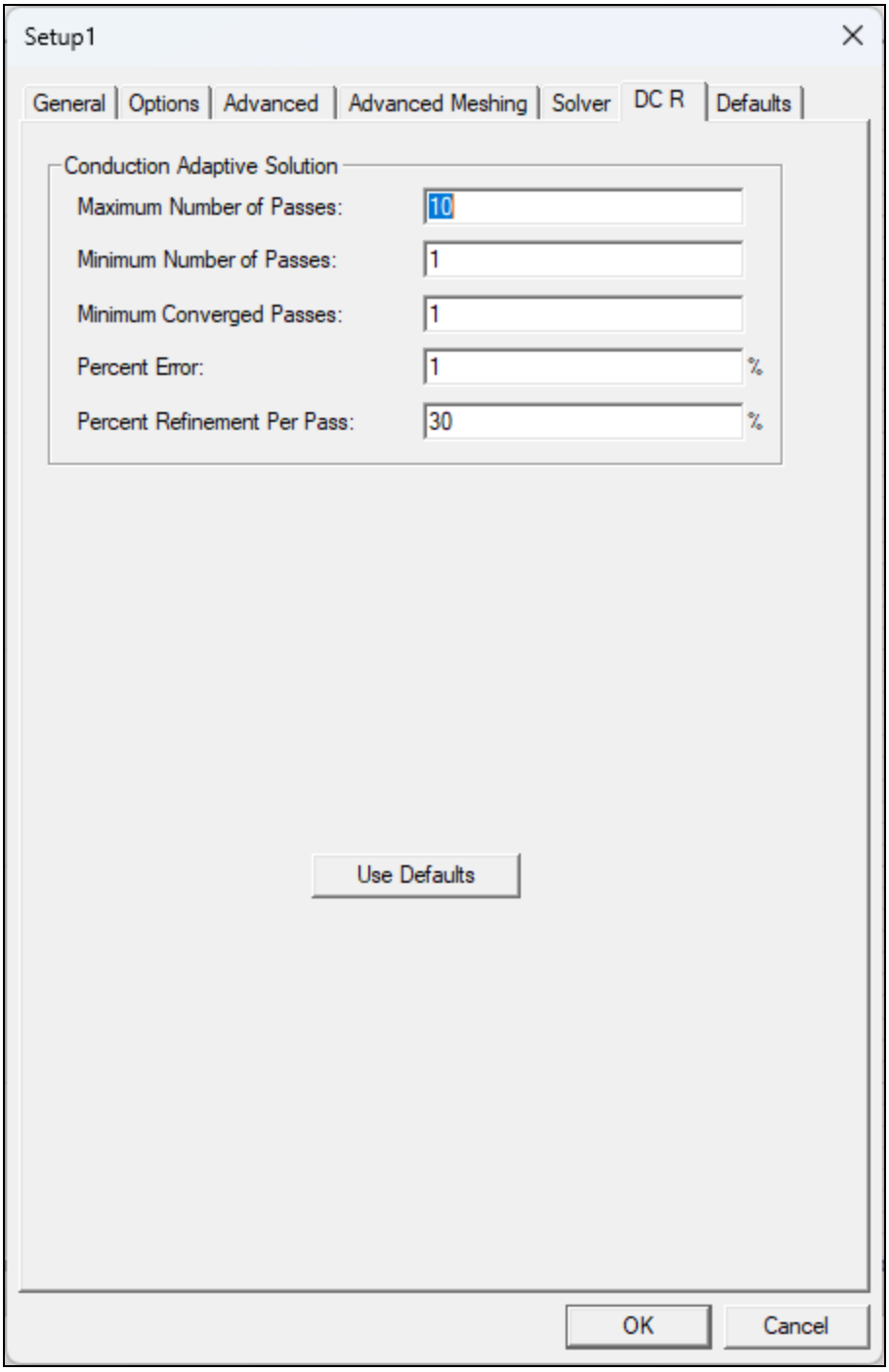
Modeling Options

- **Enable intra-plane coupling of Pwr/Gnd nets for enhanced accuracy**: Check the box to increase accuracy by including interactions between power and ground planes in simulation results.
- **Enhanced low frequency accuracy**: Check the box to increase simulation accuracy at low frequencies.

HFSS Setup: DC RTab

Select the **DC R** (i.e., DC Resistance) tab of the HFSS Setup window to access the following settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Conduction Adaptive Solution

- **Maximum Number of Passes** – the maximum number of mesh refinement cycles to perform. This value is a stopping criterion for the adaptive solution; if the maximum

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 passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis continues unless convergence criteria are reached.

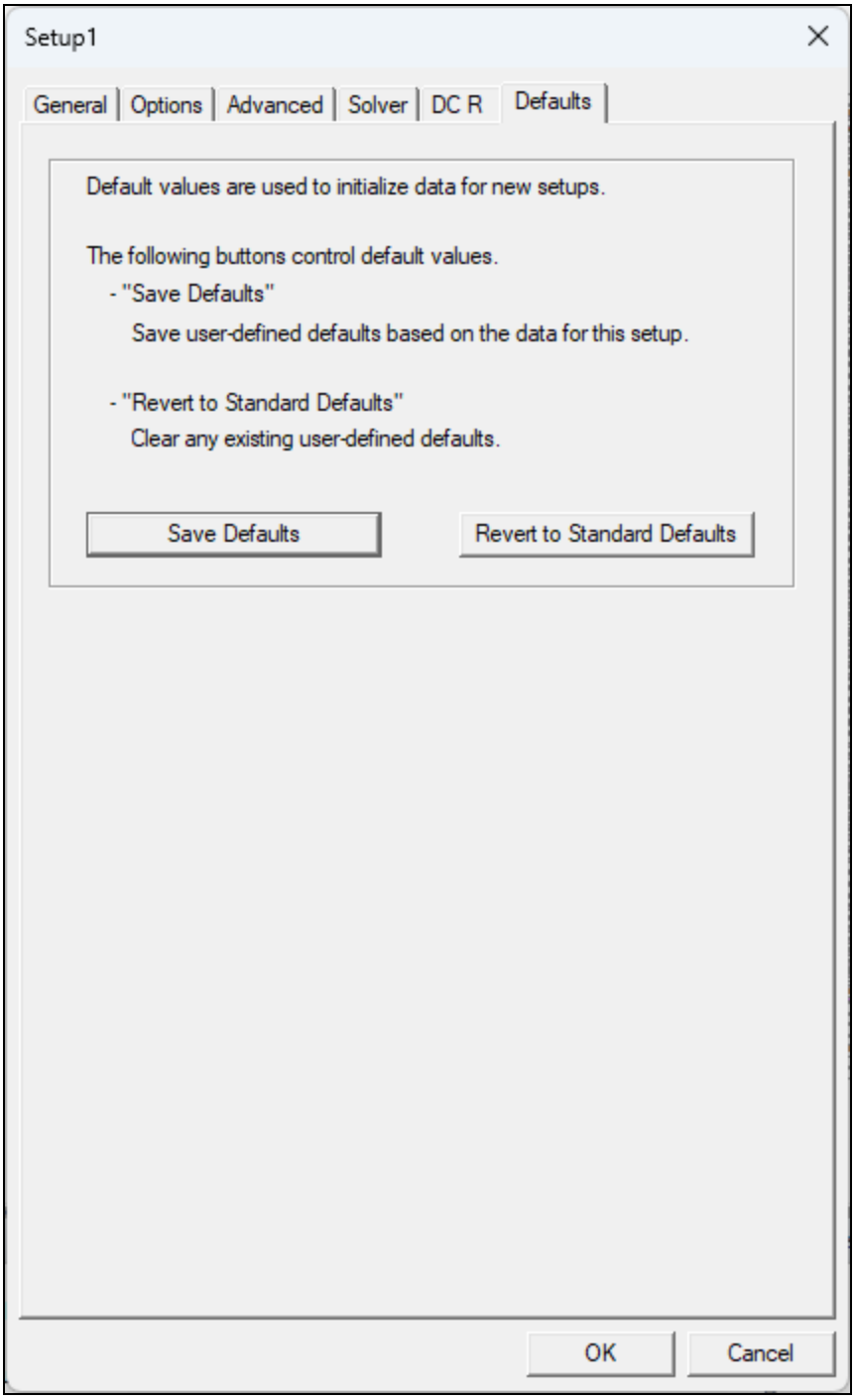
> **Important:**
>
> The size of the finite element mesh and the amount of memory required to generate a solution increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in the simulation requesting more memory than is available or taking excessive time to compute solutions.

- **Minimum Number of Passes** – the minimum number of mesh refinement cycles. The simulation will not stop analysis until after this number of passes has been completed.
- **Minimum Converged Passes** – the minimum number of passes that must meet convergence criteria before the adaptive analysis will stop.
- **Percent Error** – the desired solution accuracy. Smaller values produce more accurate but slower solutions; larger values produce less accurate but faster solutions.
- **Percent Refinement Per Pass** – determines how many tetrahedra (or prism) elements are added at each iteration of the adaptive refinement process. The elements with the highest error are refined. The default value is 30%.

HFSS Setup: Defaults Tab

Select the **Defaults** tab of the HFSS Setup window to access the following settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Save Defaults**: Save user-defined defaults based on the data for this setup for all tabs.
- **Revert to Standard Defaults**: Clear any existing user-defined defaults for all tabs.

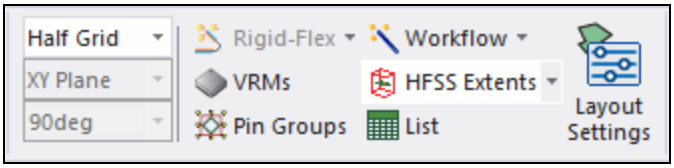
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS 3D Model Extents

The HFSS 3D Layout type can be simulated directly using HFSS. To achieve this, the bounding computational region (extents) for HFSS must be defined. The airbox's extents can have a type that is independent of the dielectric's extents.

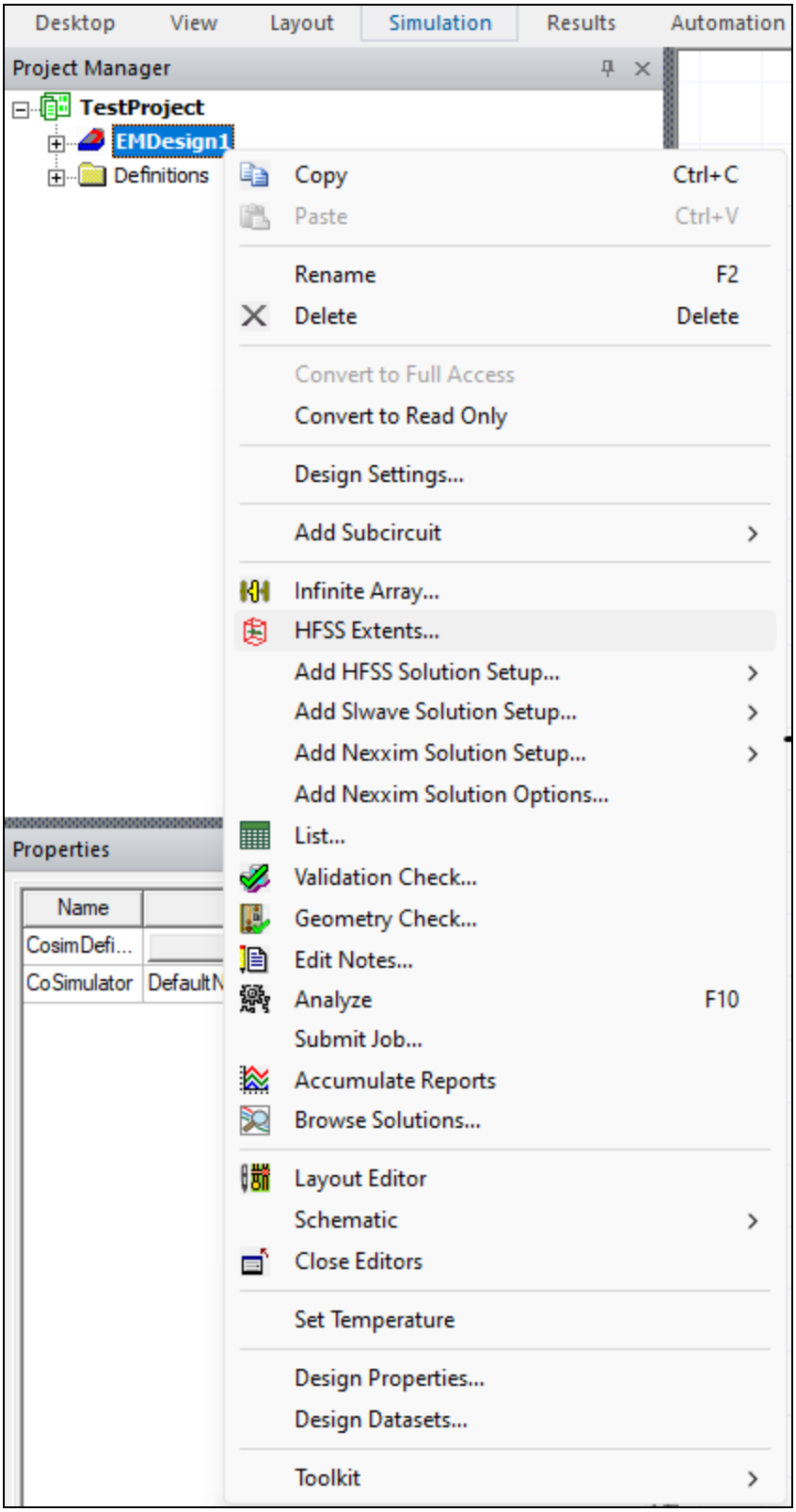
Define or Edit HFSS 3D Model Extents

1. To open the **Set HFSS Model Extents** window, do either of the following:
   - From the **Layout** ribbon, click **HFSS Extents**.



   - From the **Project Manager** window, expand the **Project Tree** (e.g., **TestProject**). Then right-click the active design (e.g., **EMDesign1**) and select **HFSS Extents**.

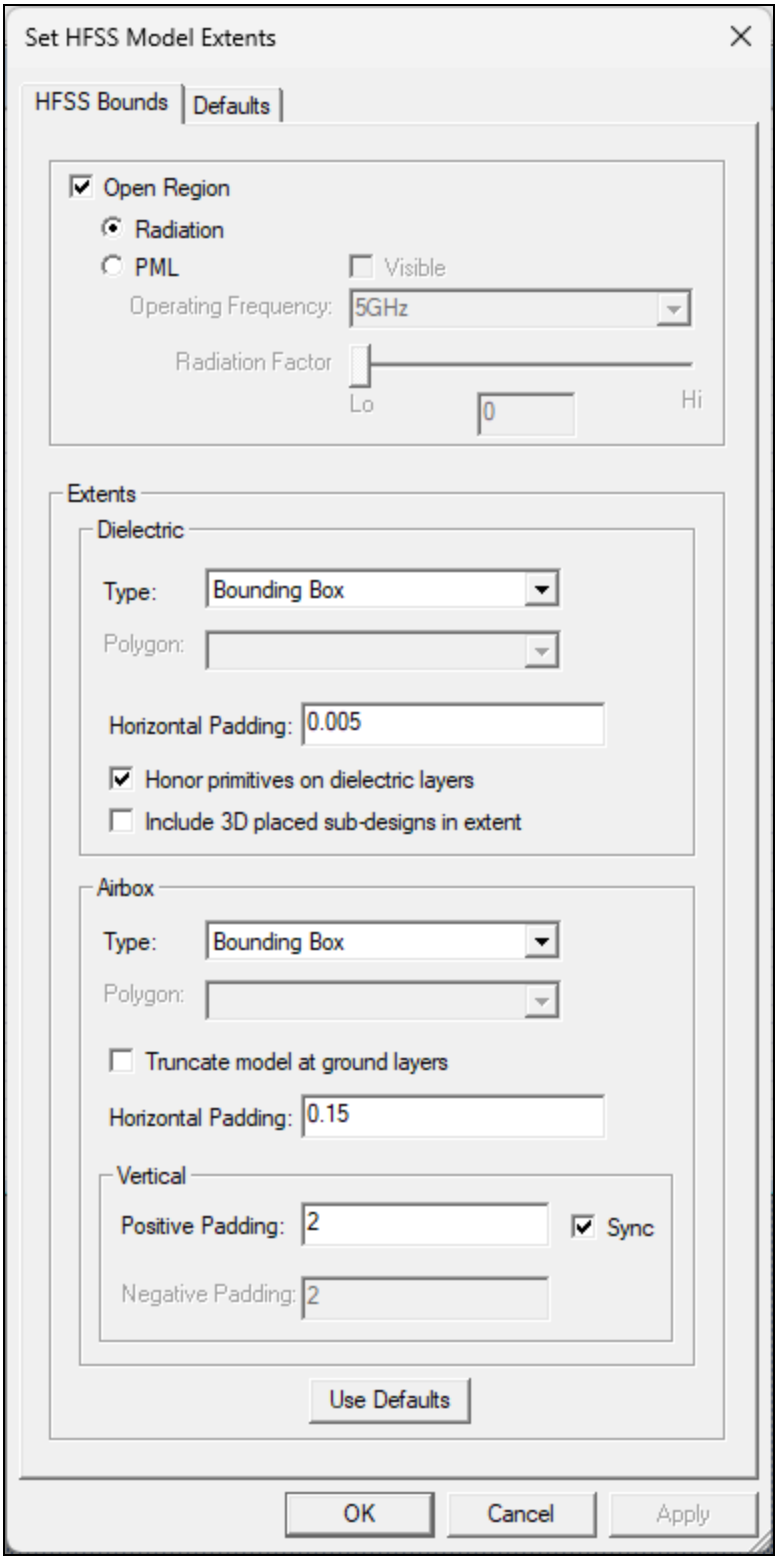
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **HFSS Bounds** tab displays 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.



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 following options:
    - **Open Region** – allows support for Radiation boundaries or perfectly matched layers (i.e., PML) boundaries in HFSS 3D Layout.
        - **Radiation** – selects a radiation open region.
        - **PML** – selects a PML open region, with the following additional options:

> **Note:**
>
> The PML option only works with bounding box extents because their geometry generation with PML has fewer corner and edge regions. When **PML** is selected:
>
> - Do **not** select **Truncate Model at Ground Layers**. It is incompatible with PML.
> - **Horizontal Padding** must be greater than zero.

            - **Visible** – sets visibility of PML boundaries.
            - **Operating Frequency** – select a frequency of PML boundary calculation from the drop-down menu and enter a value, which is automatically converted to GHz.
            - **Radiation Factor** – controls the thickness of PML boundaries, on a scale of 0 (thinnest) to 10 (thickest).
    - **Extents** – these settings control extent types and padding. **Dielectric** and **Airbox** can have different extent types:
        - **Type** – specifies the method used to derive the base-polygon for the extents region. Options include:
            - **Bounding Box** – the base-polygon is calculated as a rectangle around all active layout geometry across all layers, aligned with the X/Y axes.
            - **Bounding Box (oriented)** – the bounding box is oriented around all active layout geometry across all layers, rather than aligning with the XY axes, to create the tightest bounding box possible.
            - **Conformal** – the base-polygon is calculated as the union of all active layout geometry across all layers.
            - **Convex Hull** – the base-polygon is calculated as the convex hull of all active layout geometry across all layers.
            - Specifiy the following parameters, depending on whether they reside in the **Dielectric** or **Airbox** areas:
                - **Dielectric** – the base-polygon is defined by the exact dimensions of the zone extents input by users on the **Outline** layer. **Horizontal Padding** is deactivated and any existing value in the **Horizontal Padding** field 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.

        - **Airbox** – creates a union of the zone extent polygons on the **Outline** layer and the user-entered value in the **Horizontal Padding** field to set airbox extents.
    - **Polygon** – the base-polygon can be selected from any polygons present on user layer types. This option is only available when one or more polygons are on an assembly, outline, or user layer.
- **Polygon** – presents a list of polygons for selection when **Polygon** is the selected **Type**. The polygons must be on an assembly, outline, or user layer.
- **Dielectric** – these settings define how the polygon for dielectric and negative signal layers is generated relative to the base-polygon:
    - **Horizontal Padding** – an expansion factor (i.e., unitless fraction) or absolute offset (e.g., **0.005**, **0.15**) to be applied to the base-polygon in constructing HFSS geometry for dielectric layers.
    - **Honor primitives on dielectric layers** – If checked, and layout geometry is present on a dielectric layer, that geometry is created for the HFSS simulation. If not checked, the geometry is ignored and an expanded base-polygon is created (checked by default).
    - **Include 3D placed sub-designs in extent** – If checked, and sub-design is 3D placed, the parent design extent will include the sub-design region. If not checked, the sub-design region's geometry will not be included in the parent design extent (unchecked by default).
- **Airbox** – these settings define how the airbox is generated relative to the base-polygon:
    - **Truncate Model at Ground Layers** – if ground layers are present such that regions above or below are electromagnetically isolated from all sources, selecting this option caps the simulation volume at these elevations. Any horizontal padding on the airbox is ignored. This option is not compatible with PML.
    - **Horizontal Padding** – an expansion factor or absolute offset to be applied to the dielectric-polygon in constructing the HFSS geometry for the airbox. For PML, This must be greater than zero.
    - **Vertical** – these settings specify expansion factors for the Z-dimension:
        - **Positive Padding** – the expansion factor or absolute offset for the +Z dimension of the airbox. When applying as an expansion factor, the base dimension is the maximum of either length or width of the dielectric-polygon.
        - **Negative Padding** – the expansion factor or absolute offset for the -Z dimension of the airbox.
        - **Sync** – synchronizes **Positive Padding** and **Negative Padding** 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:** The calculations for **Airbox** extents changed in version 2020 R2: the vertical padding is calculated by multiplying the vertical padding factor to the stackup height. Previously, the vertical padding was calculated based on the horizontal extents. If a pre-2020 R2 design is opened in the current version, the vertical padding factor adjusts to keep the absolute vertical extent the same. If a design has no stackup height, its vertical padding factor is applied to the horizontal extent.

- **Use Defaults** – resets the entire window to default settings.

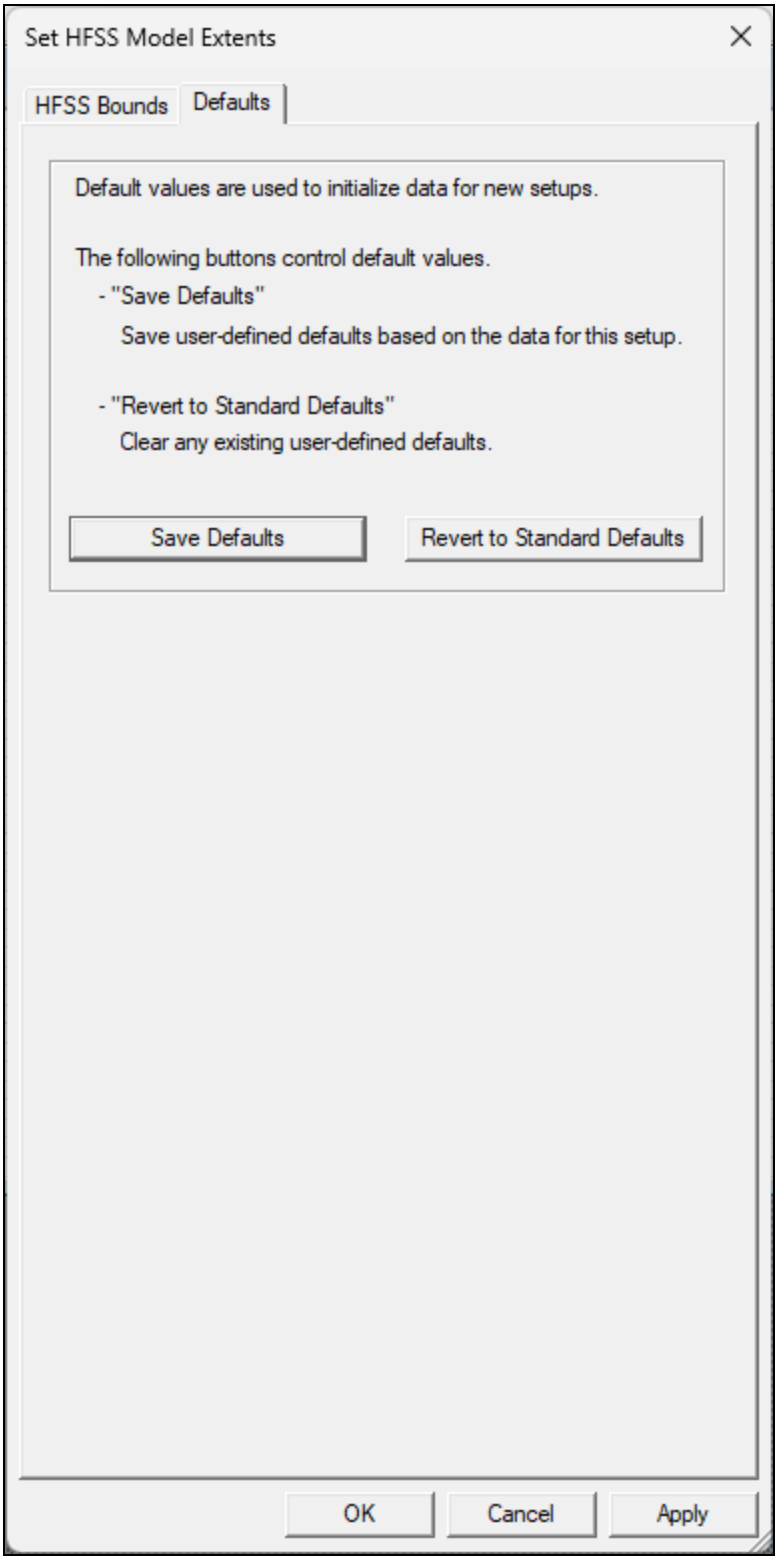
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If applicable, do the following to set the defaults:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the **Defaults** tab to display 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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Chose a default option:
   - **Save Defaults** – saves user-defined defaults based on the data for this setup (for all tabs).
   - **Revert to Standard Defaults** – clears any existing user-defined defaults (for all tabs).

4. To analyze the existing HFSS simulation setup to obtain information about the PML materials in the results project folder, click **OK**.

View or Hide the Extents

To view the extents, either click **Layout** > **Draw HFSS Air Box** on the menu bar, or on the **Layout** tab, click the down arrow adjacent to **HFSS Extents** > **Show**. A bounding box appears in the **Layout Editor**. To hide the bounding box, select **Layout** > **Draw HFSS Air Box**, or select **Hide** on the **HFSS Extends** drop-down menu on the **Layout** tab.

> **Note:** Dynamic updates of conformal HFSS extents can adversely affect Layout performance. You can improve performance by setting **Type** to **Bounding Box** in the **Set HFSS Model Extents** window or by hiding the extents.

Adding an HFSS-PI Solution Setup

HFSS-PI is a simulation workflow and solver technology for power-integrity (PI) analysis that uses prism-based meshing to efficiently handle complex on-chip, interposer, and package-level designs. It supports optional geometry simplification, such as reducing stackups, vias, and other primitives, to enable high-efficiency simulation, and it leverages large-scale high-performance computing (HPC) resources for faster and more accurate results.

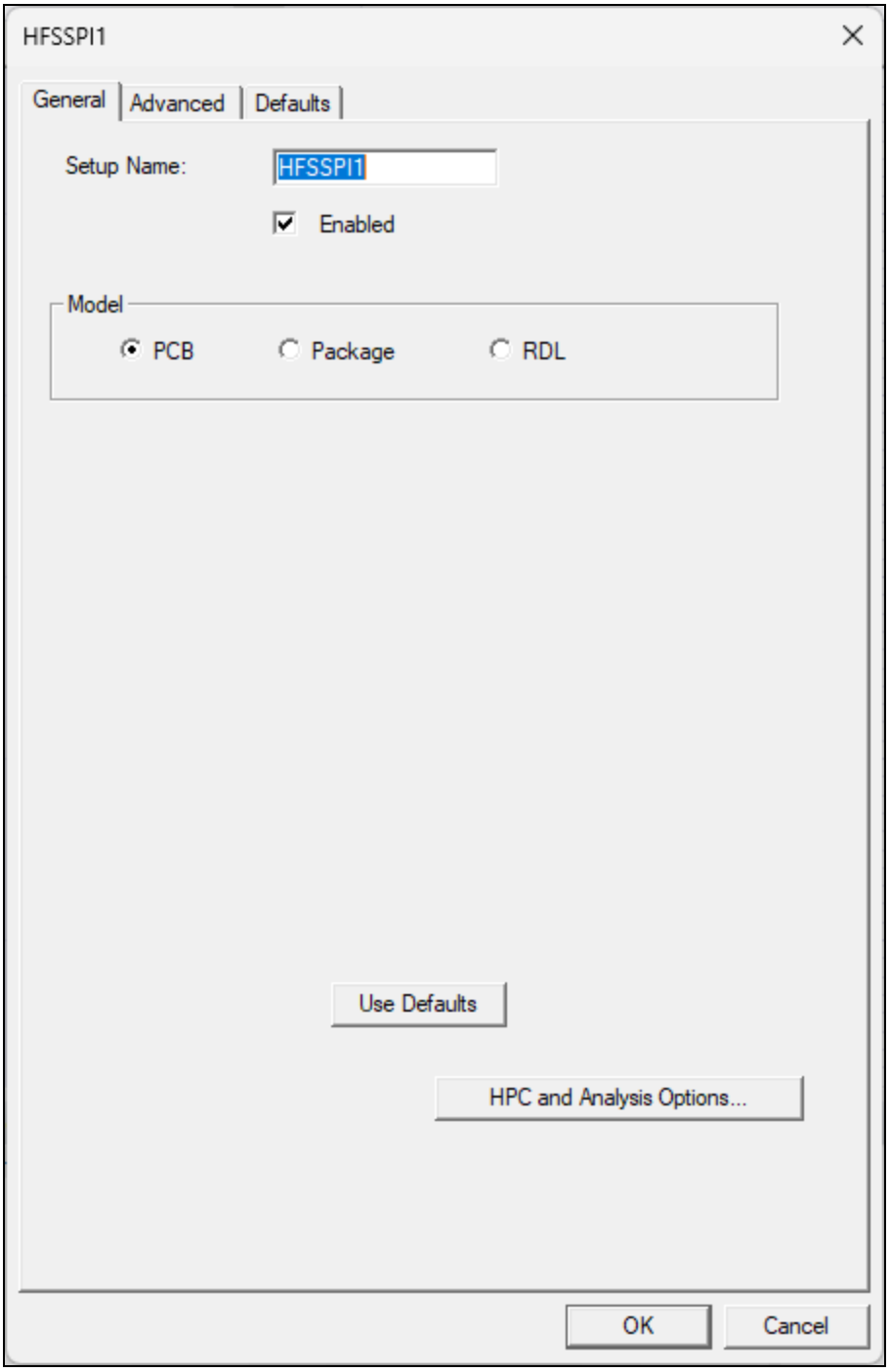
Setting Up an HFSS-PI Simulation

Follow these steps to add a HFSS-PI solution setup to a project in.

1. Open the **HFSSPI1** window. There are several ways. For example:
   - From the **Simulation** tab, click **SIwave-PSI**.
   - From the **Project Manager** window, expand the **Project Tree** > [***Active Design Folder***]. Then right-click **EMDesign1** or **Analysis** and select **Add HFSS-PI Solution Setup**.
   - From **HFSS 3D Layout**, select **Solution Setup** > select **Add HFSS-PI Solution**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup.



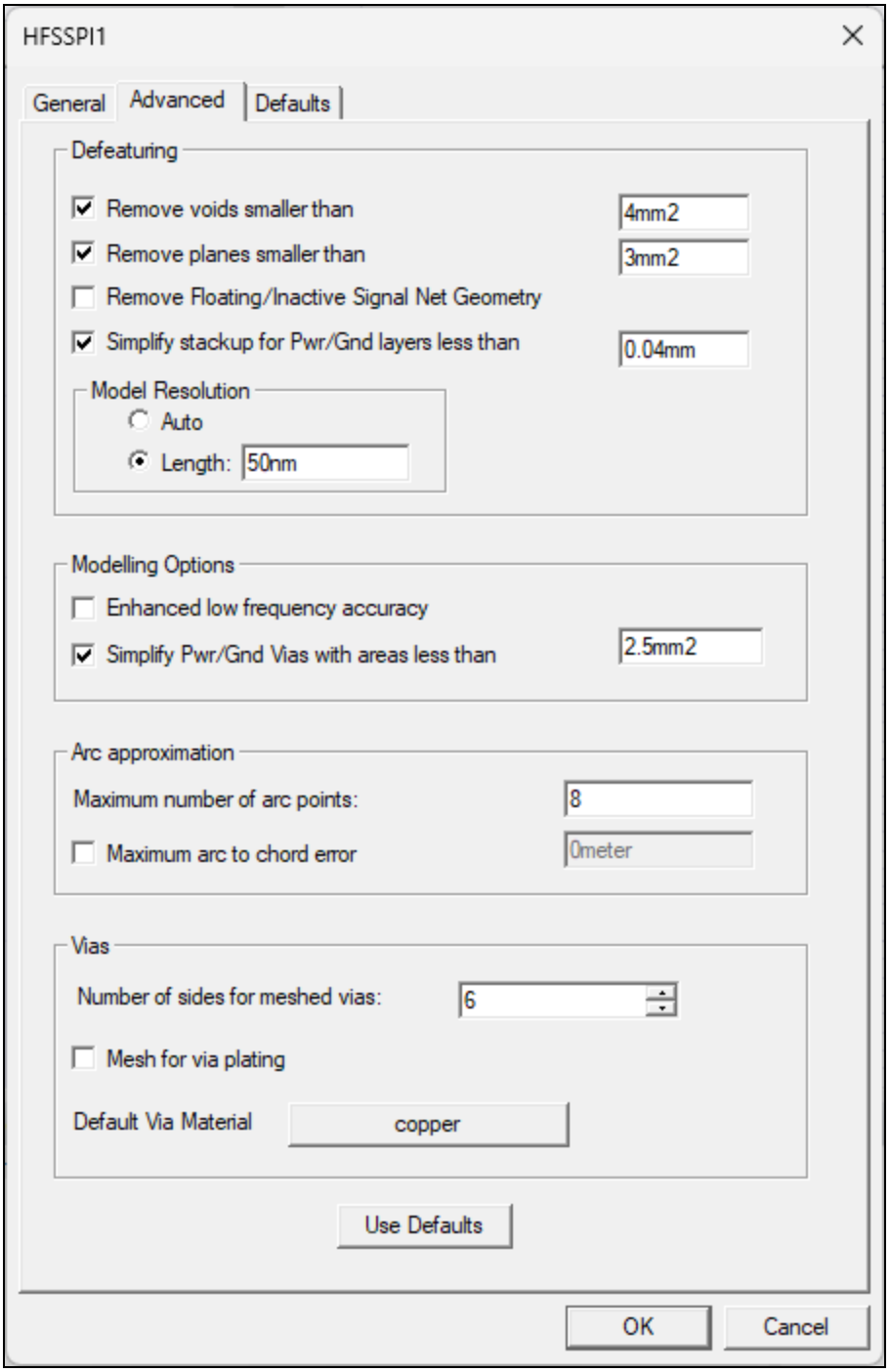
2. From the **General** tab, set the following:
   - **Setup Name** – type a name in the field or accept the default.
   - **Enabled** – check the box to activate the setup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Model Type** – select **PCB**, **Package**, or **RDL**.
  - **PCB** – for full board layout.
  - **Package** – for physical enclosure and interconnects.
  - **RDL** – for integrated circuit or interposer-level analysis.
- **HPC and Analysis Options** – open a window allowing you to specify HPC options
- **Use Defaults** – undo any configuration changes and restore defaults.

3. Click the **Advanced** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



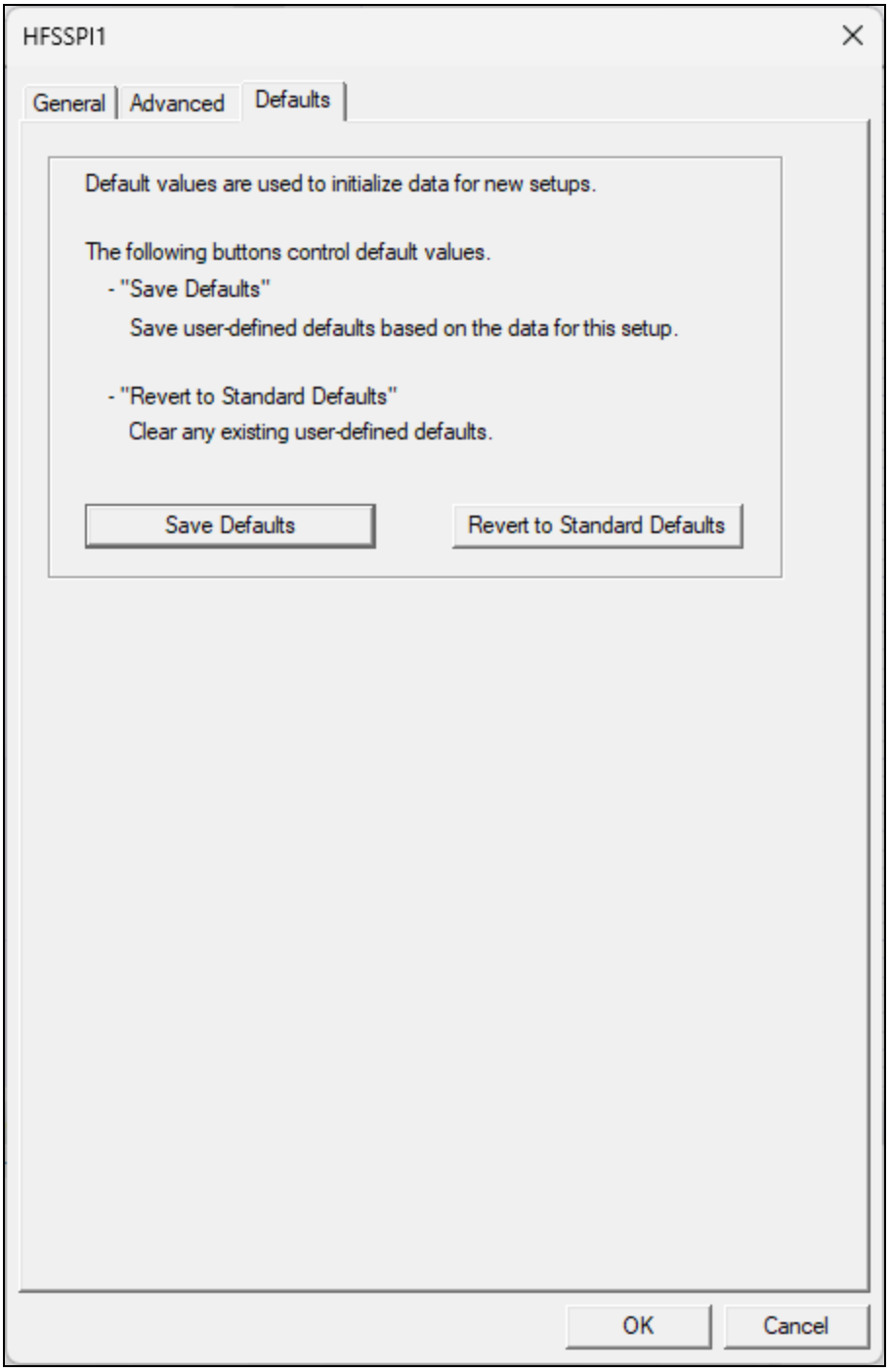
4. From the **Advanced** tab, set the following:
    - **Remove voids with an area smaller than** – check the box and enter a <value> to simplify meshing by removing small voids.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Remove Floating/Inactive Signal Net Geometry** – check the box to remove floating and inactive signal net geometry.
- **Simplify stackup for Pwr/Gnd layers less than** – check the box, enter a <value>, and metal layers with a thickness less than the specified value are set to **0** during simulation. The corresponding layer elevations are adjusted.
- **Model Resolution** – select **Auto** to automatically set the model resolution based on the design's geometry (recommended for most designs) or select **Length** to manually set a resolution threshold. Features smaller than the entered value may be merged with nearby geometry, snapped, or removed entirely, especially if they don't affect the overall structure or function of the design (recommended if the design includes fine or critical details).
- **Modeling Options**:
  - **Enhanced low frequency accuracy** – check the box to increase simulation accuracy at low frequencies.
  - **Simplify Pwr/Gnd Vias with areas less than** – check the box to increase accuracy by removing vias smaller than the specified area threshold.
- **Arc Approximation**:
  - **Maximum number of arc points** – the maximum number of mesh points on a given arc segment.
  - **Maximum arc to chord error** – check the box and enter a value for the maximum distance permissible between a straight edge approximation (the chord) and the original arc.
- **Vias**:
  - **Number of sides for mesh vias** – number of sides to use when creating the 3D representation. If less than 3, the via is modeled as a flat ribbon.
  - **Mesh for via plating** – check the box to create a mesh around the plating percent specified in the **Edit Padstack Definition** window, resulting in a more accurate DC thickness value. A via that is not fully plated includes a void filled with the material defined in the surrounding air box (e.g., air). Zero plating is not recognized and is treated as fully plated.
    - **Default Via Material** – when a material has not been specified in the padstack definition and there is no override in the properties for a via, then this is the material associated with the via mesh. It is initially set to copper.

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 the **Defaults** tab.

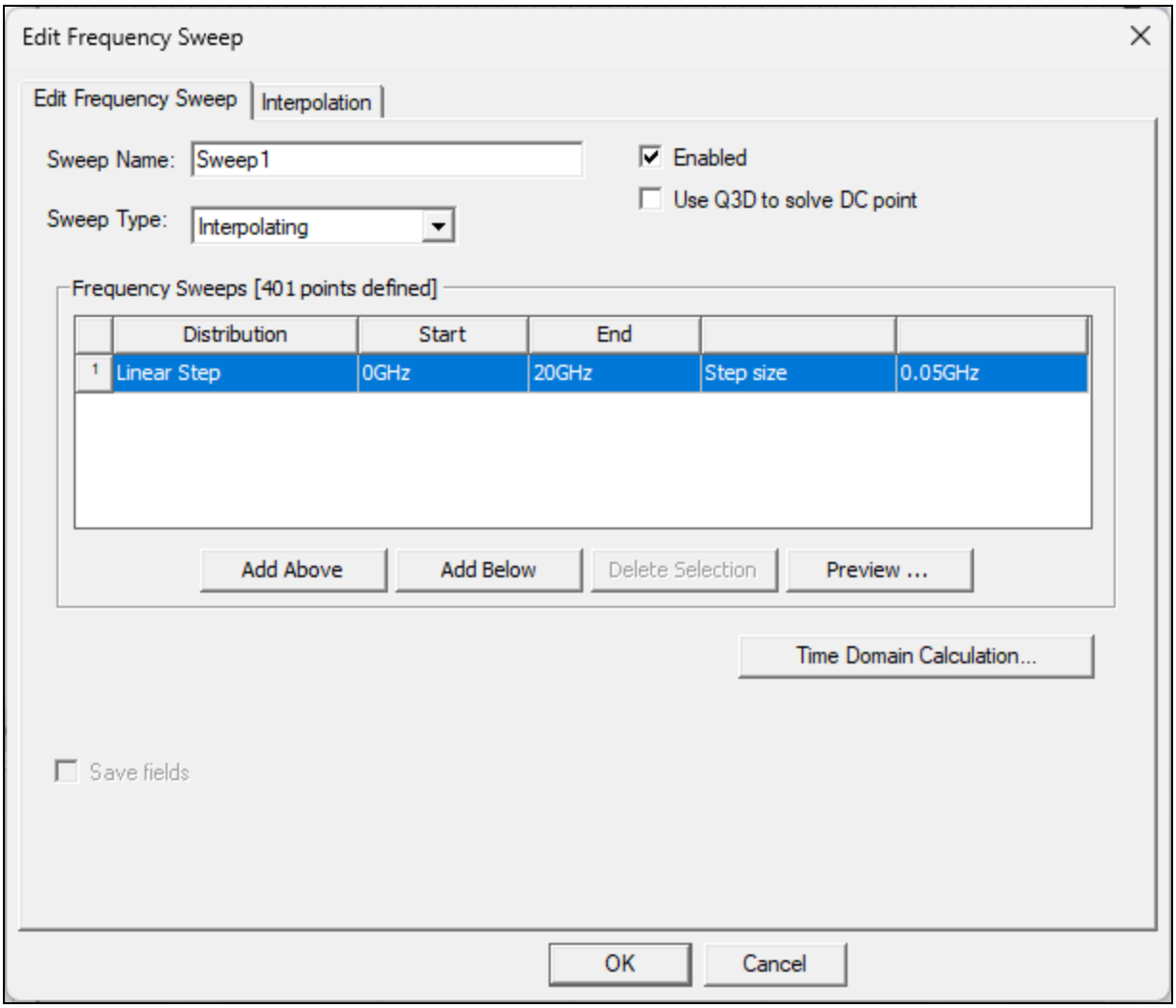


6. From the **Defaults** tab, set the following:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Save Defaults** – click to save user-defined defaults based on the data for this setup for all tabs.
- **Revert to Standard Defaults** – click to clear any existing user-defined defaults for all tabs.

7. Click **OK** to add the HFSS-PI solution setup to the active design and simultaneously open the **Edit Frequency Sweep** window.



8. From the **Edit Frequency Sweep** window, click **OK** to accept the sweep with its default values or change the default values before continuing. Refer to Adding a Frequency Sweep 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:

An enhanced interpolating sweep (i.e., enhanced PI sweep) is available for power integrity (PI) applications that require greater accuracy at low frequencies. This enhanced option is supported in both HFSS and HFSS-PI simulation setups. This enhancement is a beta feature. Beta features must be activated prior to use. Refer to Activating the Enhanced PI Sweep Beta Feature for instructions.

Analyzing the Design

To run an HFSS-PI solution setup, navigate to the **Project Manager** window, right-click **Analysis** and select **Analyze**. Refer to Running Simulations.

Post-Processing with HFSS-PI Simulation Setups

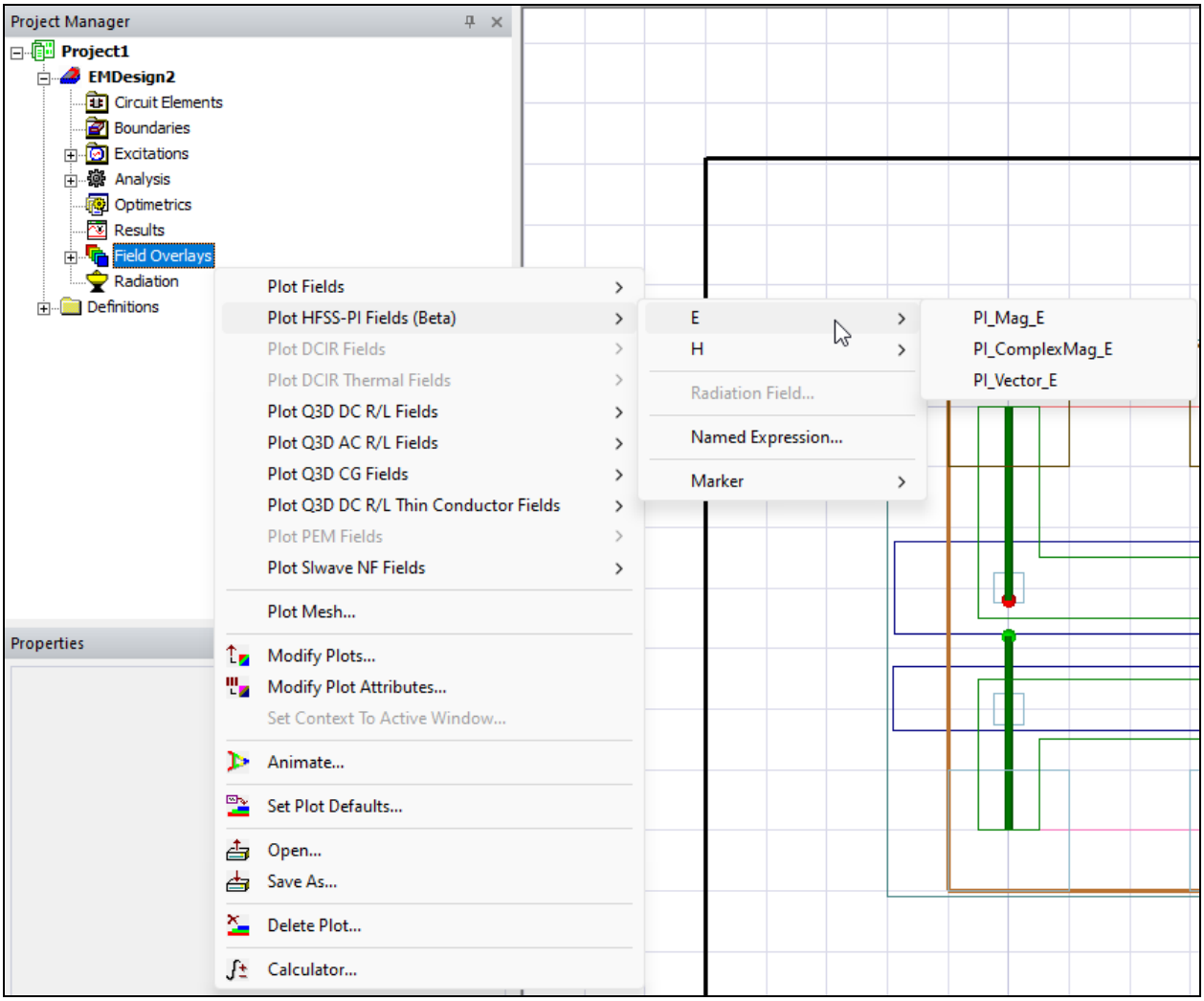
Complete these steps to plot a field overlay on the design.

Note:

HFSS-PI field plotting differs from the HFSS tetrahedral solver. E- and H-field quantities are available only at specific XY elevations defined by the solver, rather than on user-selected nets or stackup layers. Field overlays are therefore generated as XY cross-sections at signal-layer or clip-plane elevations, not as surface-based plots tied to geometry objects.

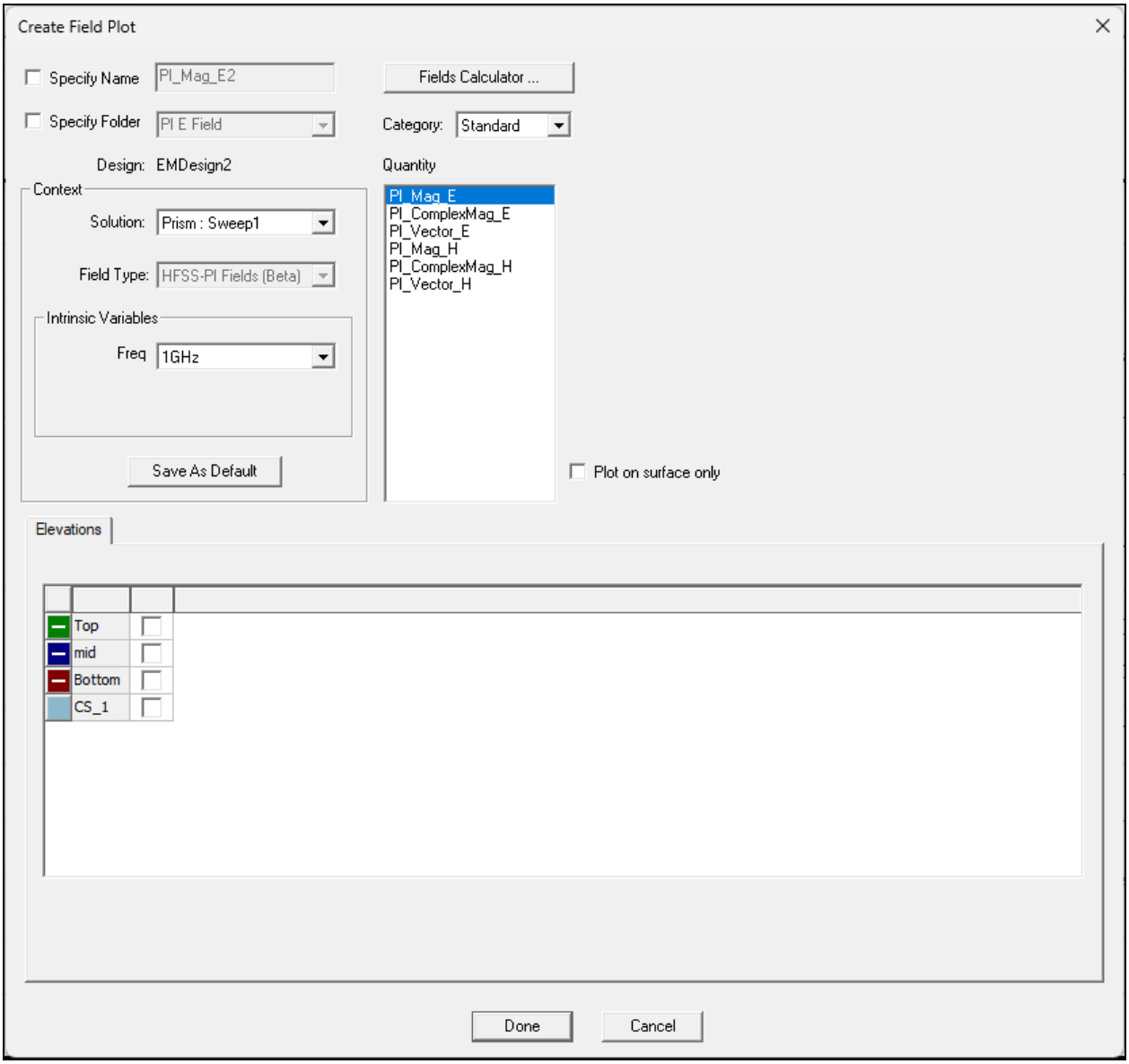
1. From the **Project Manager** window, right-click **Field Overlays** to view the available plot types. HFSS-PI solution setups provide plot types unique to the PRISM solver workflow (i.e., **PI_Mag_E**, **PI_ComplexMag_E**, **PI_Vector_E**, **PI_Mag_H**, **PI_ComplexMag_H**, and **PI_Vector_H**). These quantities support XY cross-section overlays for E- and H-fields. Refer to Field Quantities.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



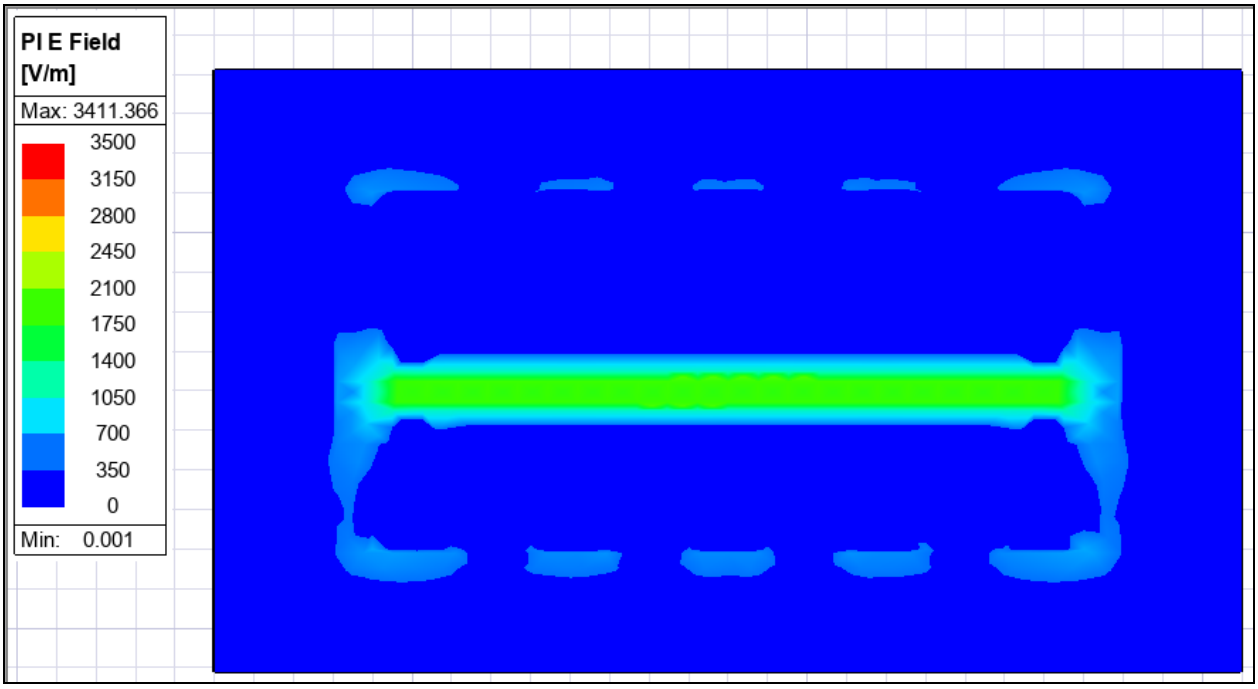
2. Select a plot type to open the **Create Field Plot** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



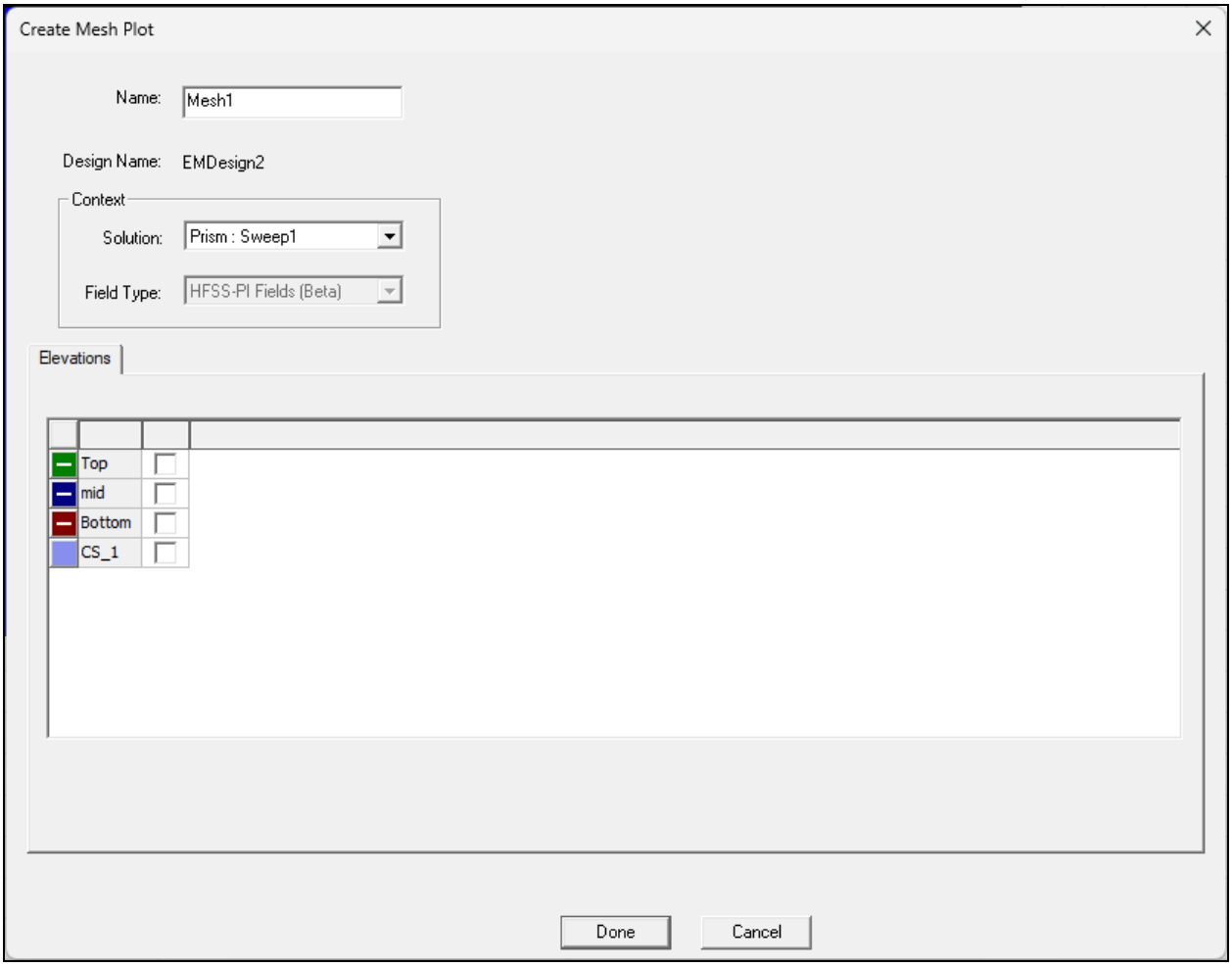
3. **Specify** a **Name**, **Folder**, and adjust the default settings in the **Context** area. From the **Quantity** list, choose one of the HFSS-PI field quantities (E-field or H-field). Then select one or more **Layers** from the **Elevations** table to define the XY cross-section to plot. Refer to Plotting Field Overlays.
4. Click **Done** to create 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.



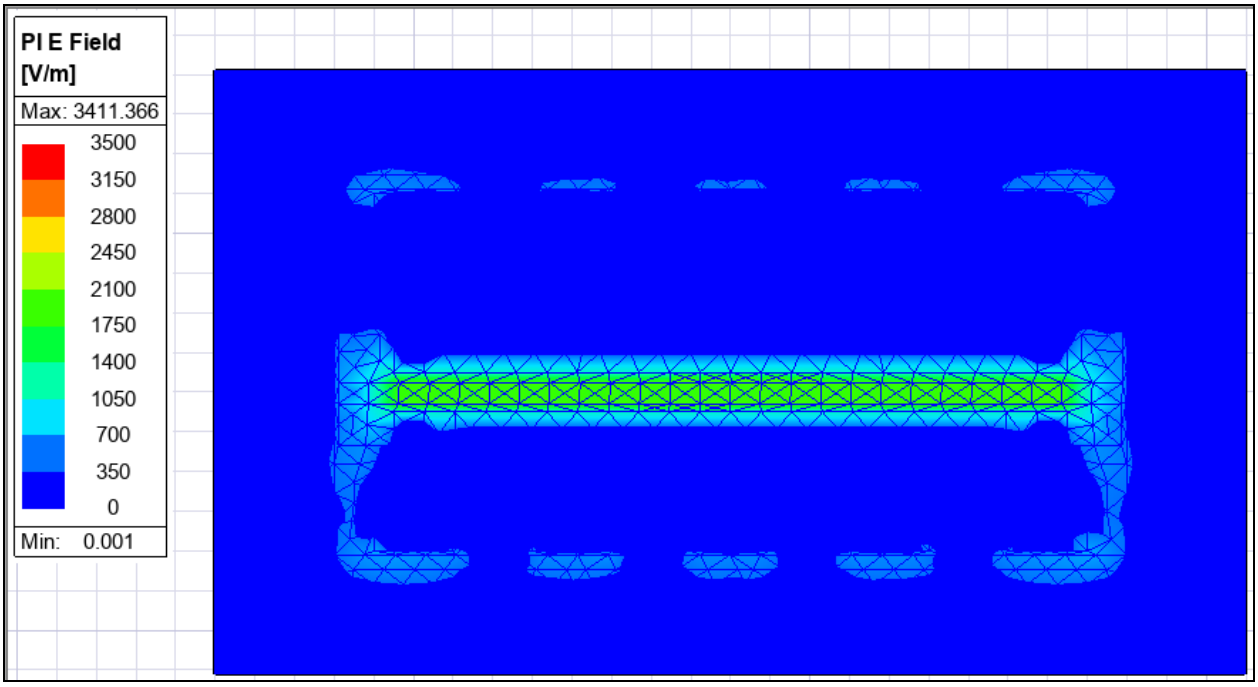
5. From the **Project Manager** window, right-click **Field Overlays** and select **Plot Mesh** to open the **Create Mesh Plot** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Select an HFSS-PI solution from the **Solution** drop-down menu to populate the **Field Type** list with PRISM-solver-specific mesh options. Then select one or more **Layers** from the **Elevations** table. Refer to Create Mesh Plot.
7. Click **Done** to create the plot mesh.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

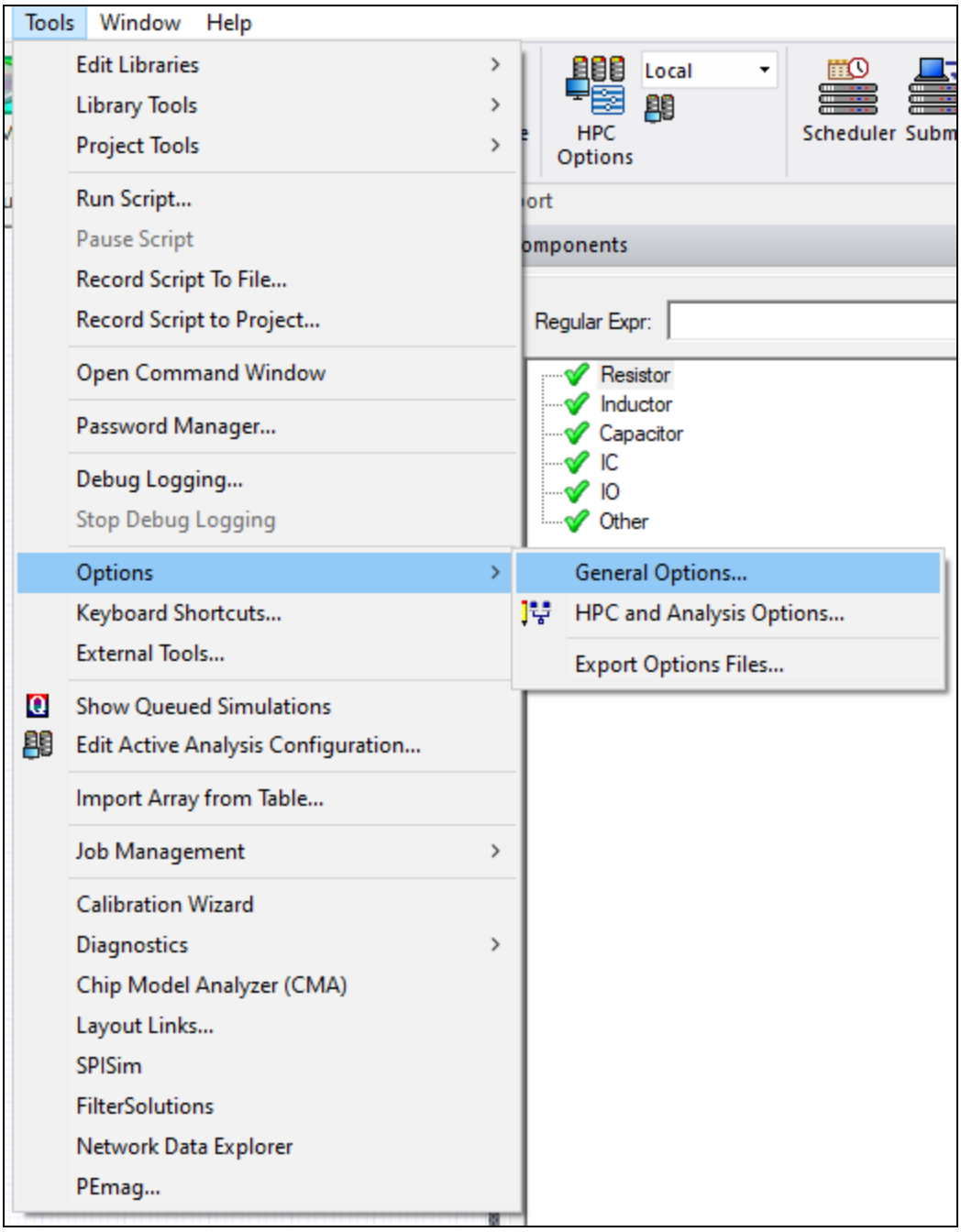


Activating the Enhanced PI Sweep Beta Feature

Follow these steps to enable the enhanced PI sweep.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

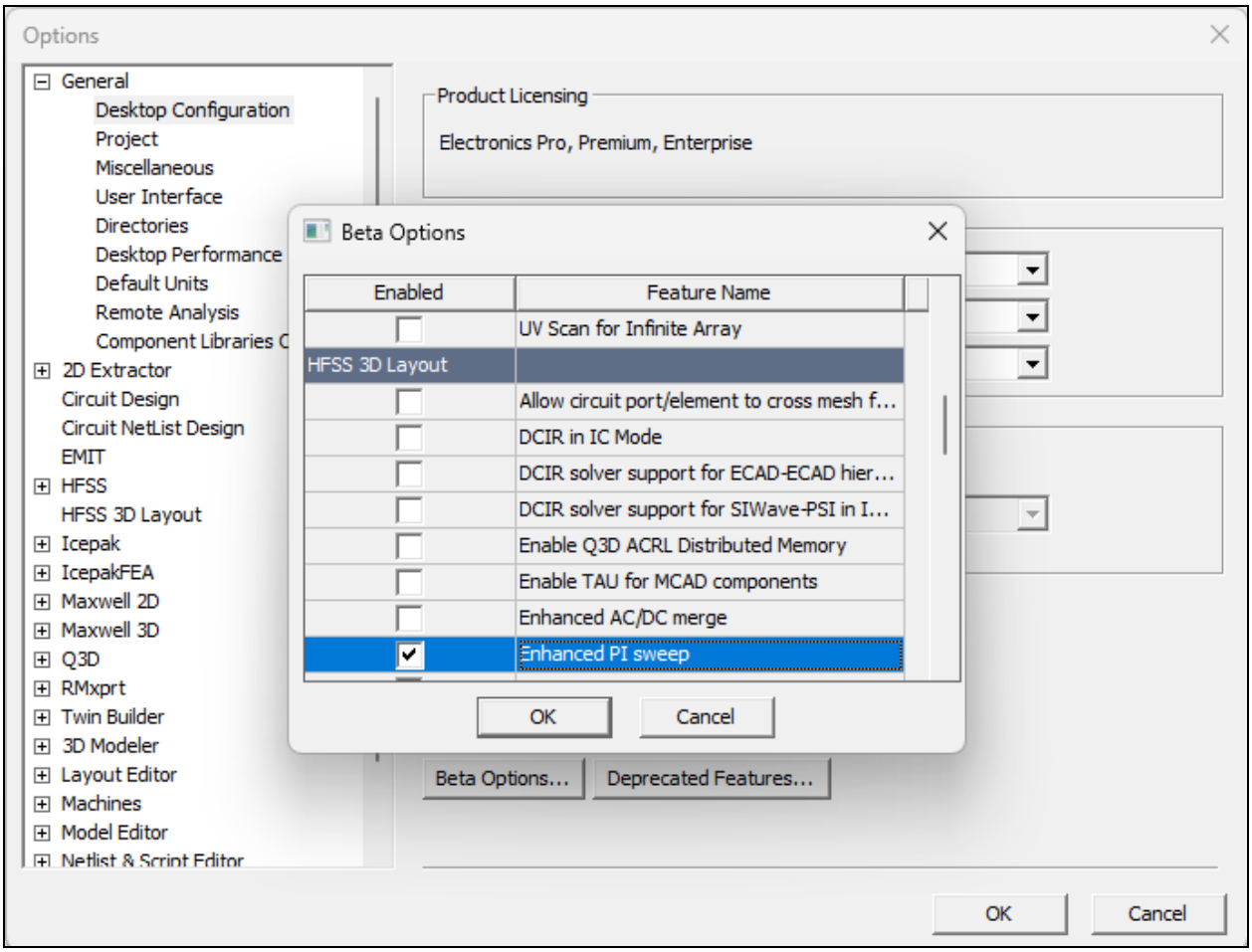
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window



2. Expand **General** > **Desktop Configuration** and click **Beta Options** to open the **Beta Options** window

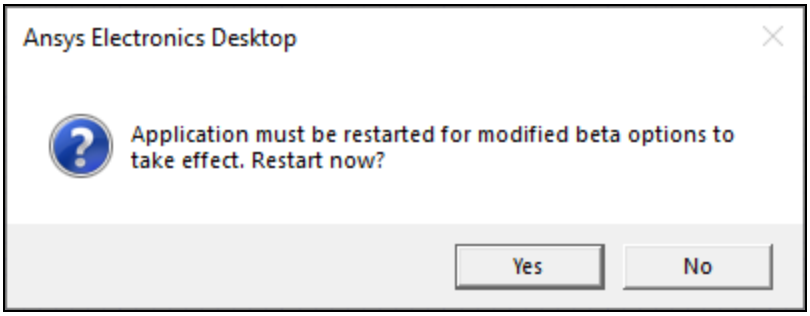
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scroll down to the **HFSS 3D Layout** section and check the box adjacent to **Enhanced PI Sweep**.



4. Click **OK** to close the **Beta Options** window.
5. Click **OK** to close the **Options** window and open a prompt to restart the software. Users may be asked to save their work.

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 **Yes** to restart immediately.
7. After restarting the software, the enhanced PI sweep will be activated. There will be no visual UI change.

The procedure is complete.

Adding a RaptorX Solution Setup (IC Mode Only)

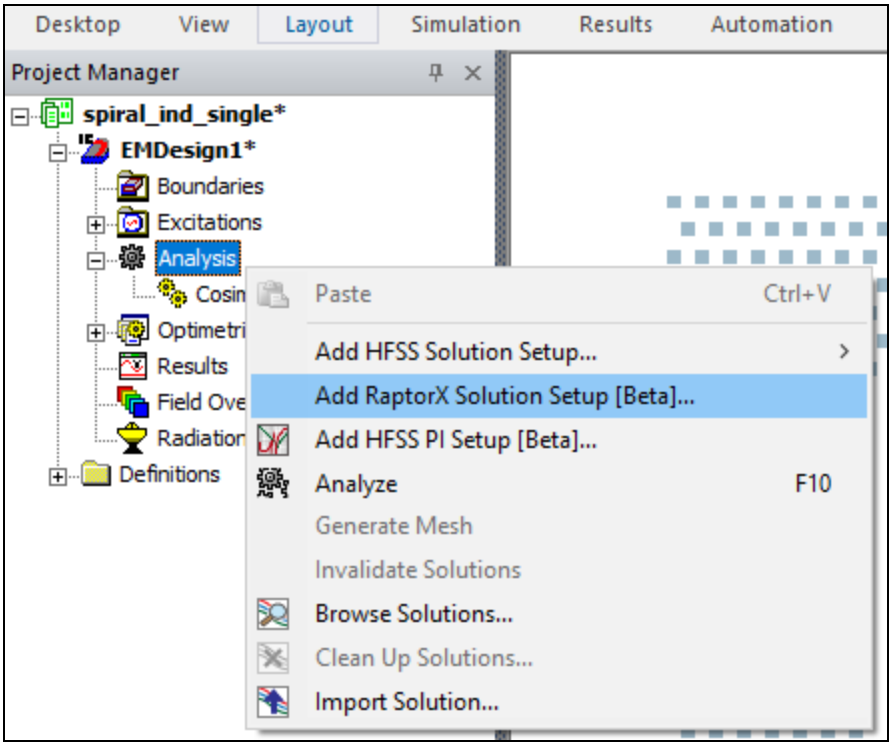
RaptorX is a pre-LVS electromagnetic (EM) modeling solver created especially for integrated circuit designs.

Setting Up a RaptorX Simulation

Follow these steps to add a RaptorX solution setup to a project in IC mode.

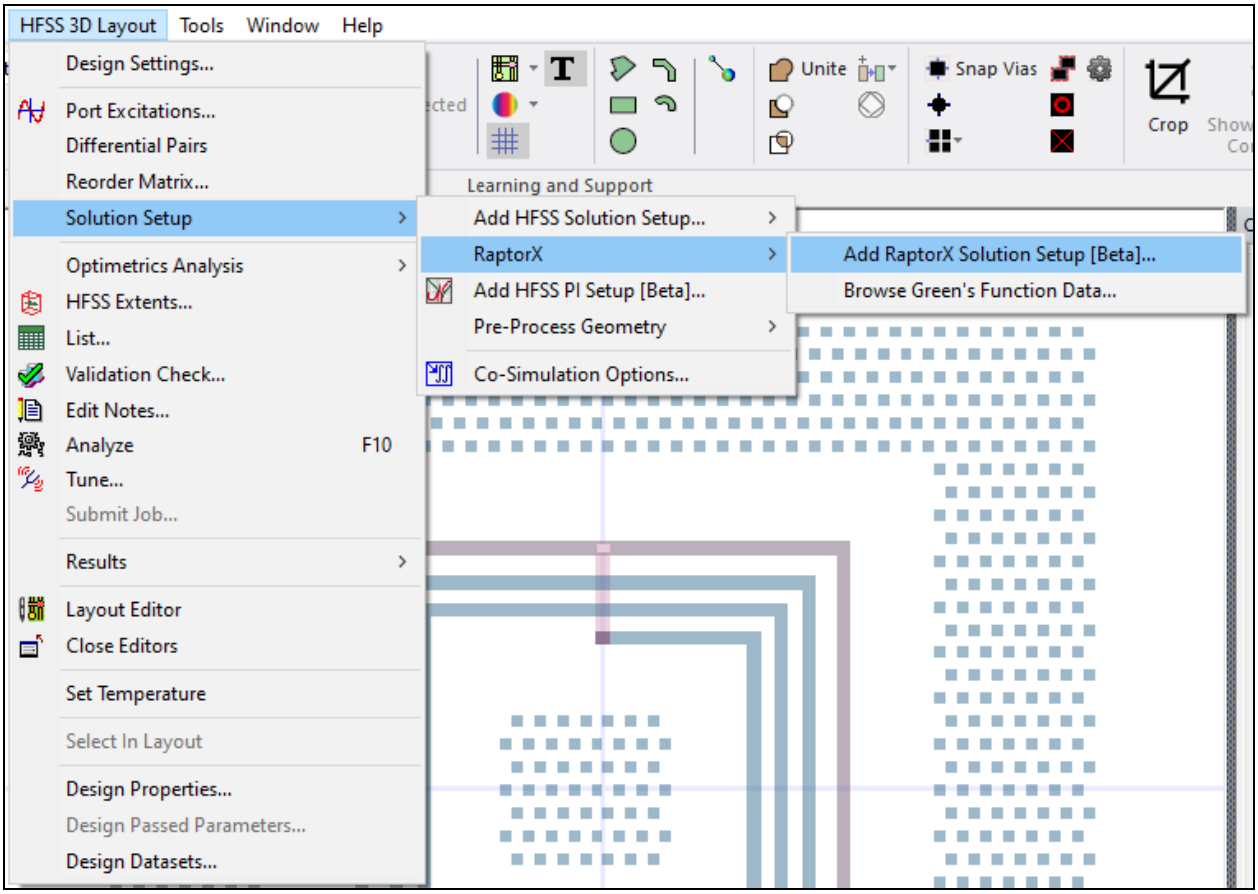
1. Open the **RaptorX Solution Setup** window by doing one of the following:
   - From the **Project Manager** window, expand the **Project Tree** > [*Active Design Folder*] (e.g., **spiral_ind_single** > **EMDesign1**). Then right-click **Analysis** and select **Add RaptorX Solution Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



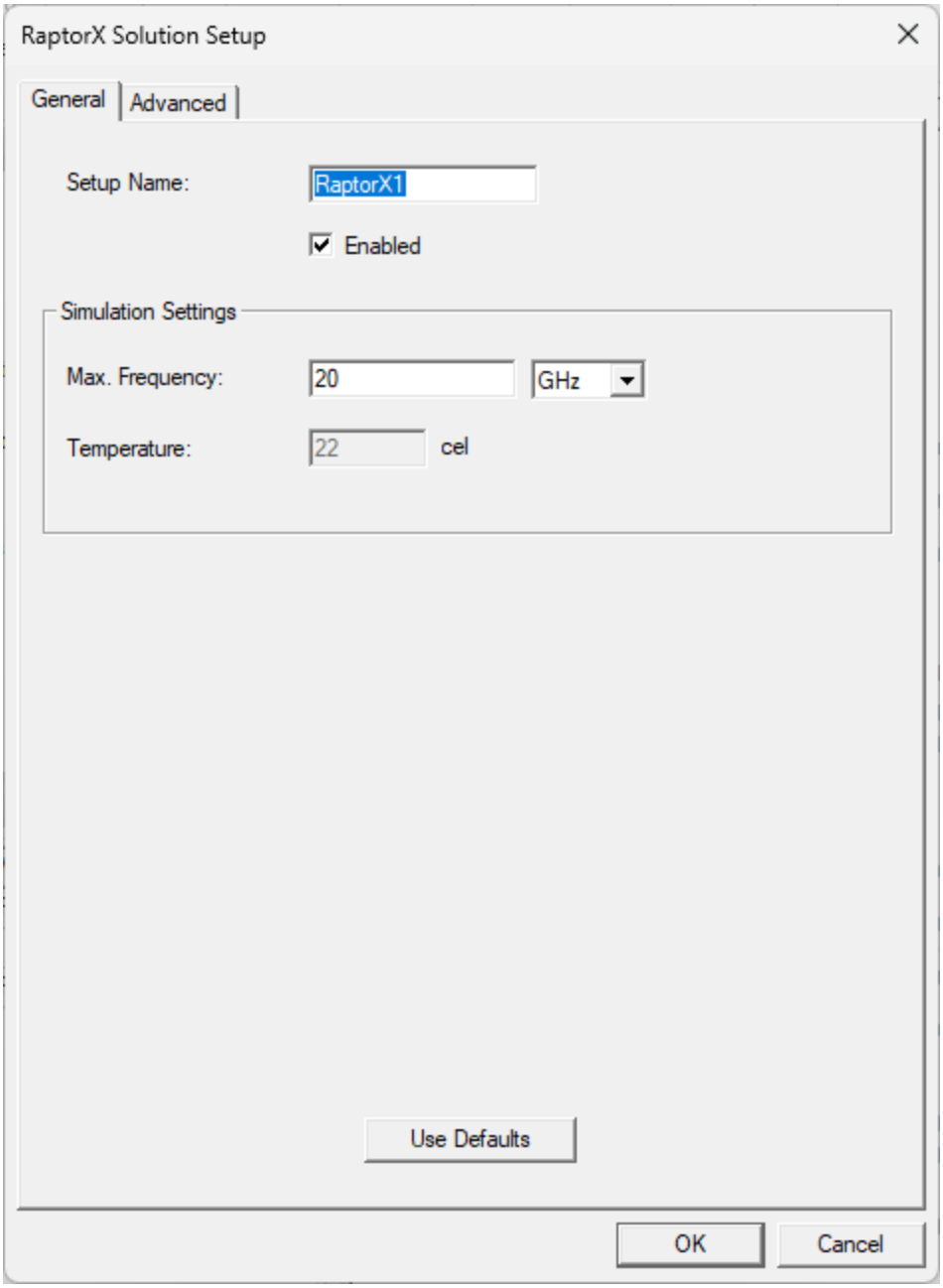
- From **HFSS 3D Layout**, select **Solution Setup** > **RaptorX** > **Add RaptorX Solution Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **RaptorX Solution Setup** window, make selections from the **General** and **Advanced** tabs. Default 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.

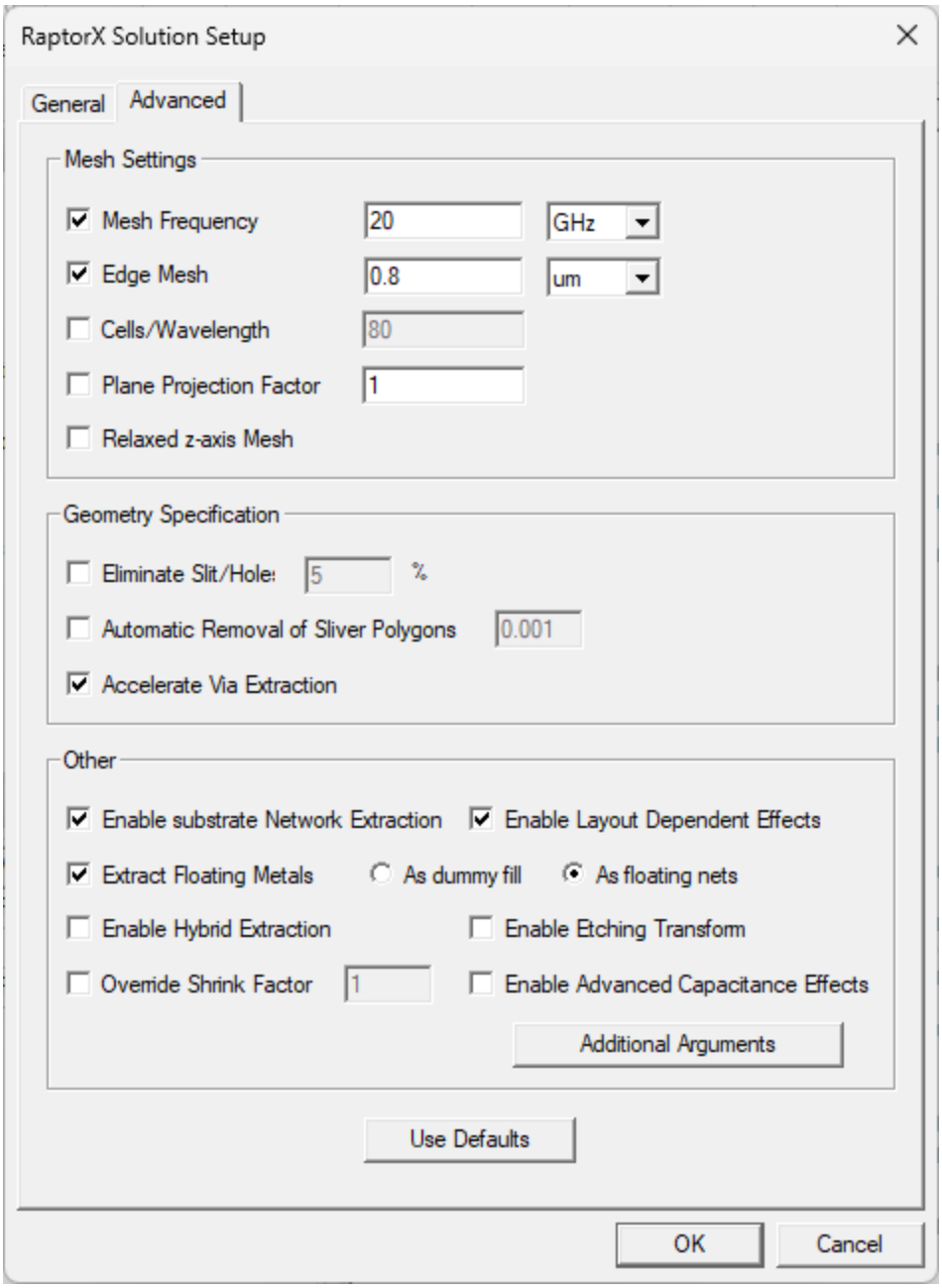


3. From the **General** tab, set the following:
    - **Setup Name** – type a name in the field or accept the default.
    - **Enabled** – check the box to activate the setup.
    - **Max. Frequency** – enter a value in the provided field to specify the maximum simulation frequency (Ansys recommends at least 10 GHz) and select a unit of measurement from the corresponding 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.

- **Temperature** – specifies the simulation temperature. To modify the simulation temperature, refer to Setting Uniform Temperature in HFSS 3D Layout.

4. Click the **Advanced** tab.



5. From the **Advanced** tab, check the corresponding box next to any or all of the following settings to activate them. Then enter any appropriate value or make drop-down menu selections, as applicable.

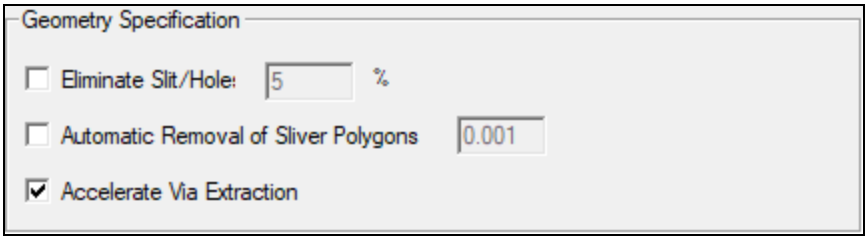
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. From the **Mesh Settings** area, set the following:

- **Mesh Frequency** – enter a value in the provided field to specify a custom frequency for mesh generation. Then select a unit of measurement from the corresponding drop-down menu. A higher mesh frequency improves simulation accuracy.
- **Edge Mesh** – enter a value in the provided field to specify the thickness and width of the exterior conductor filament. Then select a unit of measurement from the corresponding drop-down menu. **Edge Mesh** prevails over **Max. Frequency** and **Mesh Frequency**. If **Edge Mesh** is left unchecked, edge mesh is calculated internally equal to the layer depth defined by **Mesh Frequency** or **Max. Frequency**.
- **Cells/Wavelength** – enter a value in the provided field to specify the number of cells under each wavelength. Wavelength is calculated according to **Max. Frequency** or **Mesh Frequency** (e.g., If users enter **20** in the **Cells/Wavelength** field, the design is divided into 10 cells if its width or length is 1/2 wavelengths).
- **Plane Projection Factor** – enter a value in the provided field to set a hard limit on the complexity of the simulation. The mesh conforms to the limit to improve extraction time. Activate **Plane Projection Factor**, then refer to Changing Net Settings to set individual values for certain nets.
- **Relaxed z-axis Mesh** – check the box to simplify mesh along the z-axis. If the thickness of a conductor is greater than three times the layer depth, then instead of performing dynamic meshing, **Relaxed z-axis Mesh** creates no more than three filaments along the z-axis. The top and bottom filament thickness are equal to layer depth, as previously calculated, and the middle filament is equal to the remaining thickness. This improves extraction time and resource use with a negligible effect on accuracy.

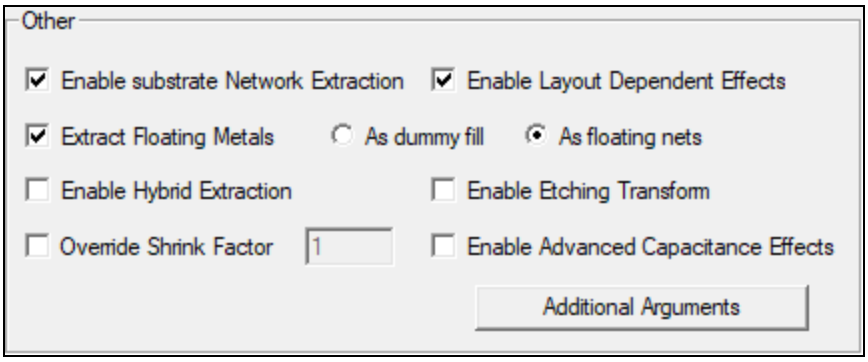
b. From the **Geometry Specification** area, set the following:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Eliminate Slit/Holes** – enter a value (i.e., threshold) to simplify designs with strain relief or thermal relief slits and holes. Any slits/holes below the threshold are filled.
- **Automatic Removal of Sliver Polygons** – enter a value threshold to simplify layouts by aligning slightly misaligned polygons. Any overlap below the threshold is removed.
- **Accelerate Via Extraction** – check the box to merge neighboring vias. Merged vias are treated as ideal (i.e., non-resistive).

c. From the **Other** area, set the following:



- **Enable substrate Network Extraction** – models substrate coupling effects by simulating an equivalent distributed RC network. Checked by default.
- **Enable Layout Dependent Effects** – calculates the variation of resistivity as a function of a conductor's drawn width and spacing to its neighbors or as a function of its local density, due to dishing, slotting, cladding thickness, et cetera. Checked by default.
- **Extract Floating Metals** – after activating, select either of the following:
  - **As dummy fill** – calculates the effective capacitance between any pairs of metal segments in the design, while taking into account the presence of dummy metal islands that are not included in a net. This fill is represented as a new entity (i.e., *dummyFill*). **Mesh Frequency** does not apply to *dummyFill*.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **As floating nets** – groups every metal segment in the design not explicitly included in a net in a new entity (i.e., *floatingNets*), which will be meshed according to **Mesh Frequency** settings.
- **Enable Hybrid Extraction** – assumes electromagnetic (EM) effects dominate the top (metal) layers of the design, so the design is separated horizontally into two parts. The upper part is assumed to contain a smaller number of ports compared to the bottom, potentially decreasing extraction time significantly.
- **Override Shrink Factor** – enter a value to override the default shrink factor.
- **Enable Etching Transform** – distorts the design based on foundry rules and applies the conductor's bias (i.e., positive/negative, deflation/inflation) at the conductor's edges due to presumed unavoidable effects in the manufacturing process. Ignored if LDE data is not present.
- **Enable Advanced Capacitance Effects** – applies capacitance-related effects to the design (e.g., conformal dielectrics, loading effects, and/or dielectric damage).
- **Additional Arguments** – click to open the **Additional Extraction Arguments** window. From the **Additional Extraction Arguments** window, do the following:
    a. Enter a parameter in the **Name** field and an appropriate **Value**. Select from one or more of the following:

Argument Name	Value
pe-edt-package-vias	Comma-separated list of via layers, which will be designated as package vias (e.g., **via1**, **via2**, et cetera). **Note:** This parameter is ignored if a technology definition is applied to the design.
pp-circloid-arc-simplification-min-no-of-points-in-arc	The minimum number of points in a circloid arc. Default value is **5**.
pp-circloid-arc-simplification-try-hexadecagons	• **True** (Default) – Use hexadecagons for circloid-arc simplification. • **False** – Do not use

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

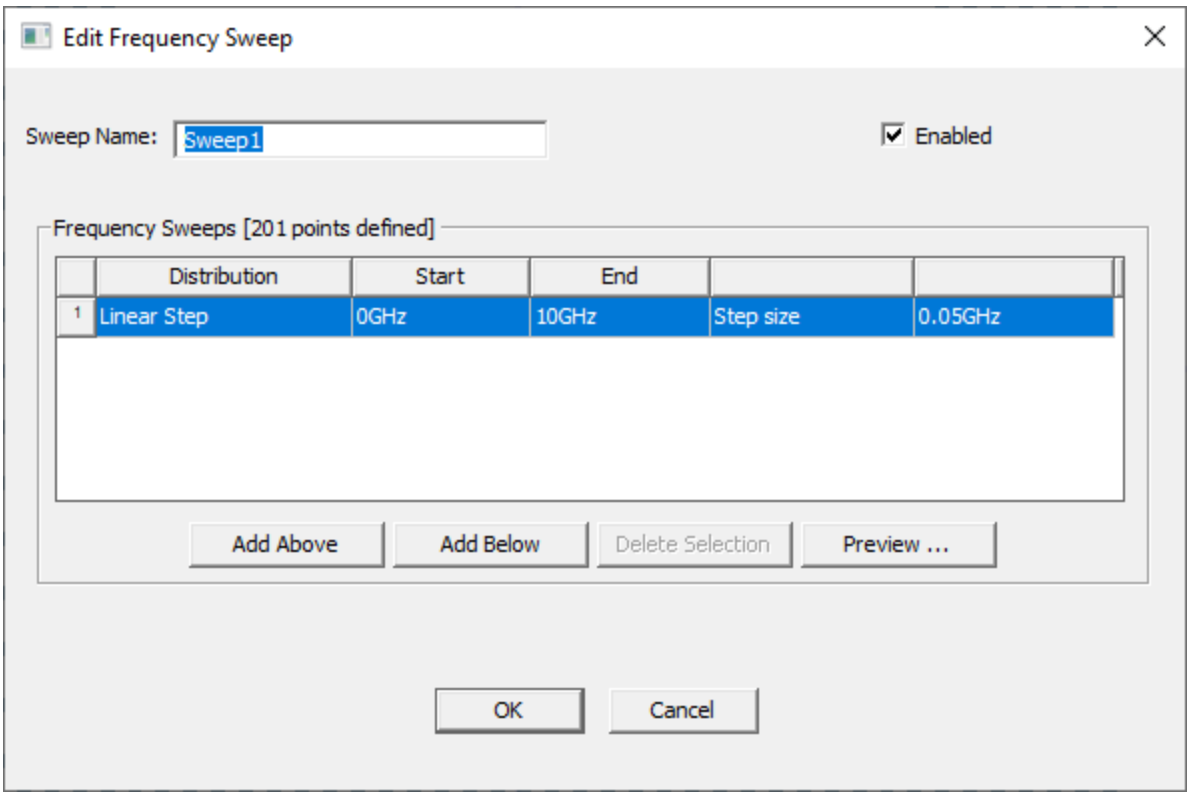
	hexadecagons for circloid-arc simplification. **Note:** If **False**, then **pp-circloid-arc-simplification-try-octagons** (if inserted) must have the value **True**.
pp-circloid-arc-simplification-try-octagons	• **True** – Use octagons for circloid-arc simplification. • **False** (Default) – Do not use octagons for circloid-arc simplification. **Note:** If **False**, then **pp-circloid-arc-simplification-try-hexadecagons** (if inserted) must have the value **True**.
pp-via-stacking	• **True** (Default) – A stacking approach is used on silicon vias, merging any overlapping vias in adjacent layers into a single stacked region. • **False** – A stacking approach is not used on silicon vias.

b. Click **Insert** to save the argument.

c. Repeat steps **a**-**b**, as appropriate. Then click **OK** to close the window and return to the **RaptorX Solution Setup** 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. Click **OK** to add the RaptorX solution setup to the active design and simultaneously open the **Edit Frequency Sweep** window.



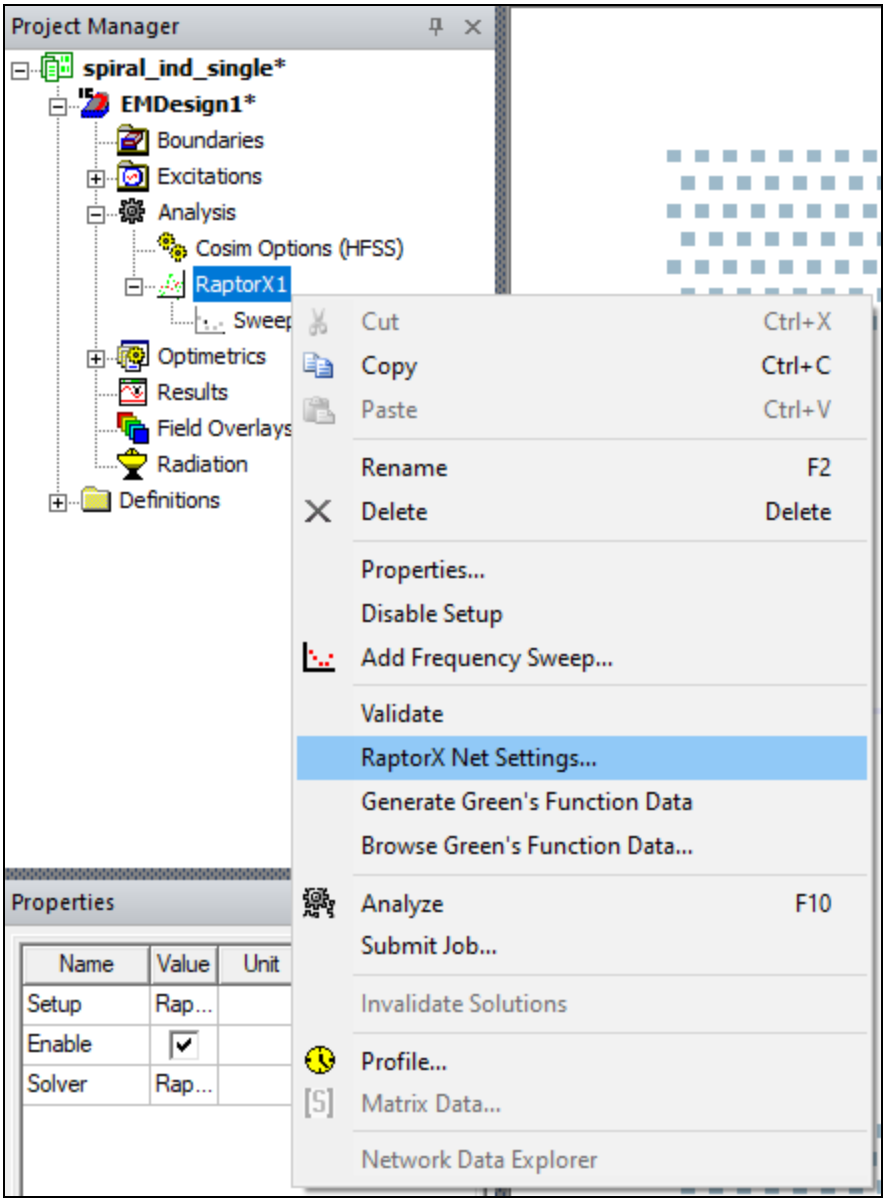
7. From the **Edit Frequency Sweep** window, click **OK** to accept the sweep with its default values or change the default values before continuing. Refer to Adding a Frequency Sweep for more information.

Changing Net Settings

RaptorX solution setups can be customized further by making changes to the **Mesh Frequency** and **Edge Mesh** parameters for specific nets. Follow these steps to view all of the nets recognized by RaptorX during simulation, including the virtual nets created as a result of user-selected options selected during setup (e.g., such as the **floatingNets** group created by activating **Extract Floating Metals**), and modify as appropriate.

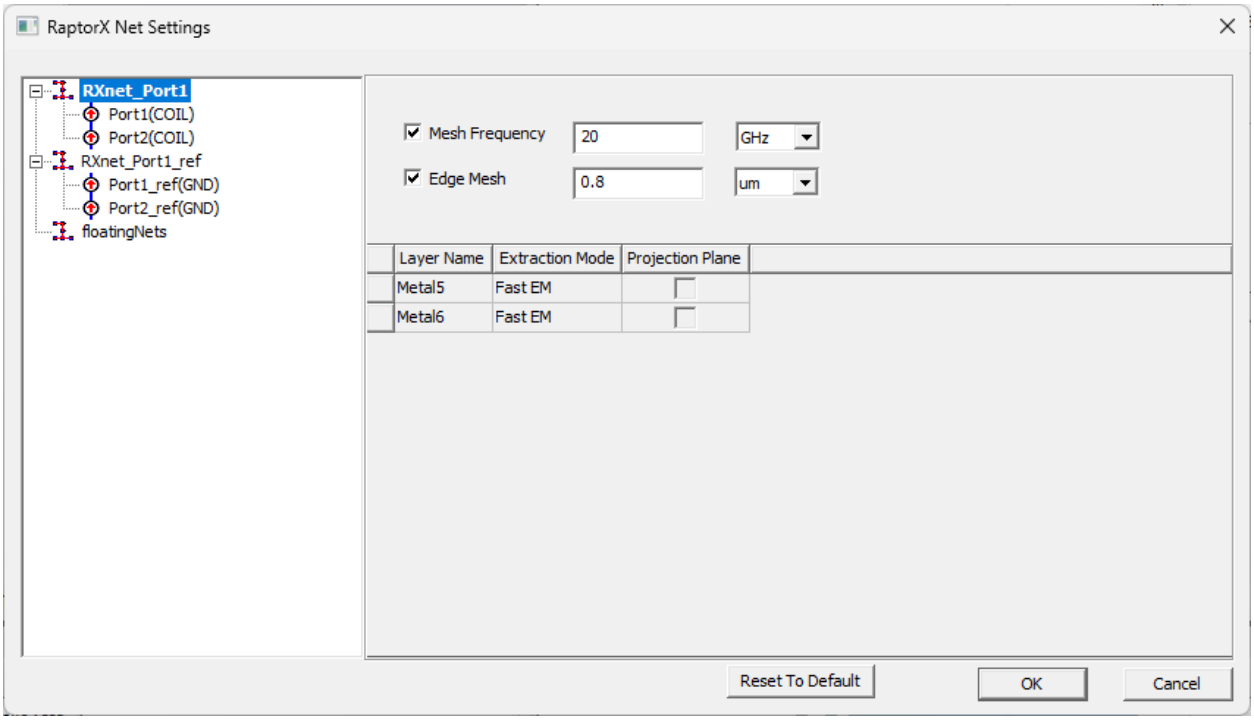
1. From the **Project Manager** window, expand **Analysis**. Then right-click the RaptorX solution setup (e.g., **RaptorX1**) and select **RaptorX Net Settings** to open the **RaptorX Net Settings** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **RaptorX Net Settings** window, select a mesh to view its individual settings or **Ctrl**+**click**/**Shift**+**click** to select multiple meshes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Check the corresponding box next to any or all of the following settings to activate them. Then enter any appropriate value or make drop-down menu selections, as applicable. The settings function as described in Setting Up a RaptorX Simulation.
    - **Mesh Frequency**
    - **Edge Mesh**
    - **Projection Plane** (active if **Plane Projection Factor** is also active in the **RaptorX Solution Setup** window)
4. Click **OK** to close the **RaptorX Net Settings** window.

Applying Technology

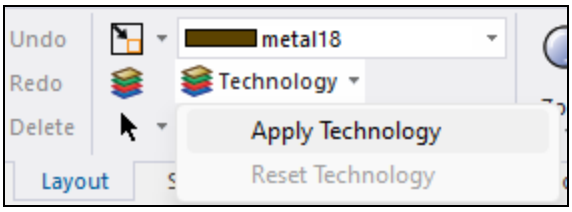
If any technology files were imported during the initial setup of the design, follow these steps to apply a technology before simulation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

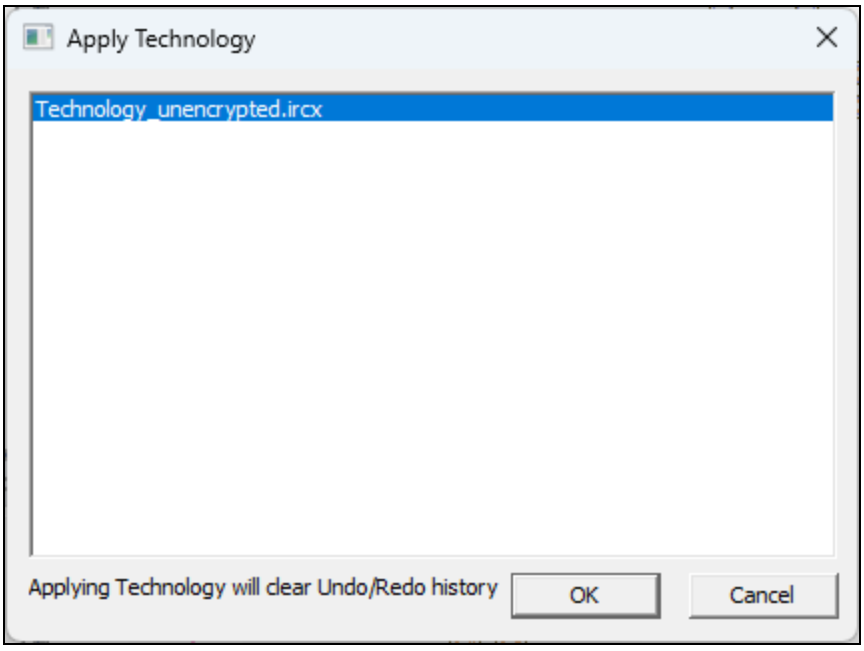
Note:

If technology files were **not** imported during the initial setup of the design, users can add one or more files at any time before simulation. If a technology file **was** applied during setup, the technology's name should already appear in the **Layout** ribbon adjacent to the **Technology** drop-down menu. Users will not need to follow these steps. Refer to Adding a Technology File to a Design.

1. If a technology is available, navigate to the **Layout** ribbon and select **Apply Technology** from the **Technology** drop-down menu.



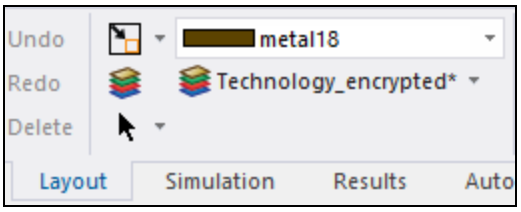
2. Once the **Apply Technology** window opens, select an appropriate technology from the list and click **OK**.



Once the technology is applied to the design, the technology's name appears adjacent 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 **Technology** drop-down menu.

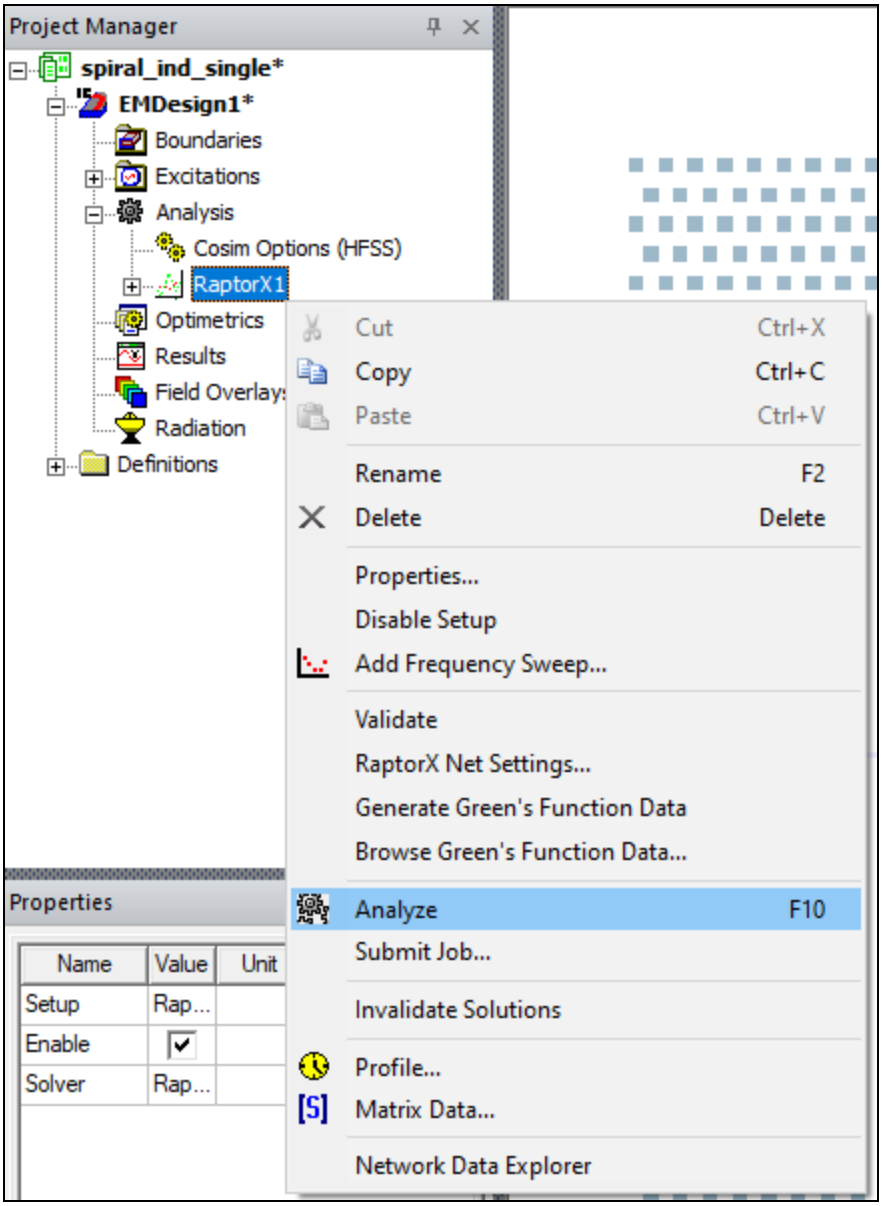


Analyzing the Design and Post-Processing

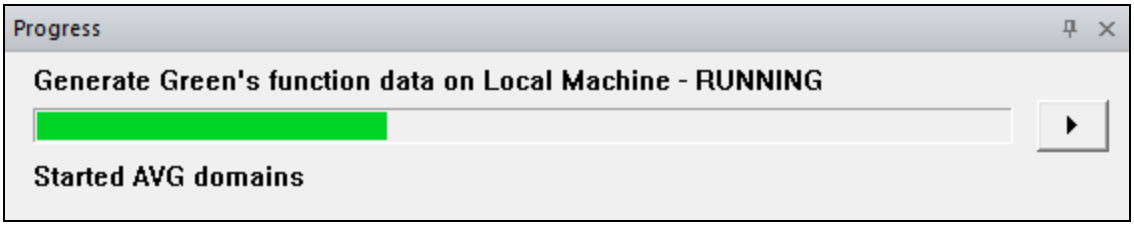
Follow these steps to analyze the new RaptorX solution setup, then create an overlay, mesh, and generate a report.

1. From the **Project Manager** window, right-click **RaptorX1** and select **Analyze** to run the simulation. Refer to Running Simulations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

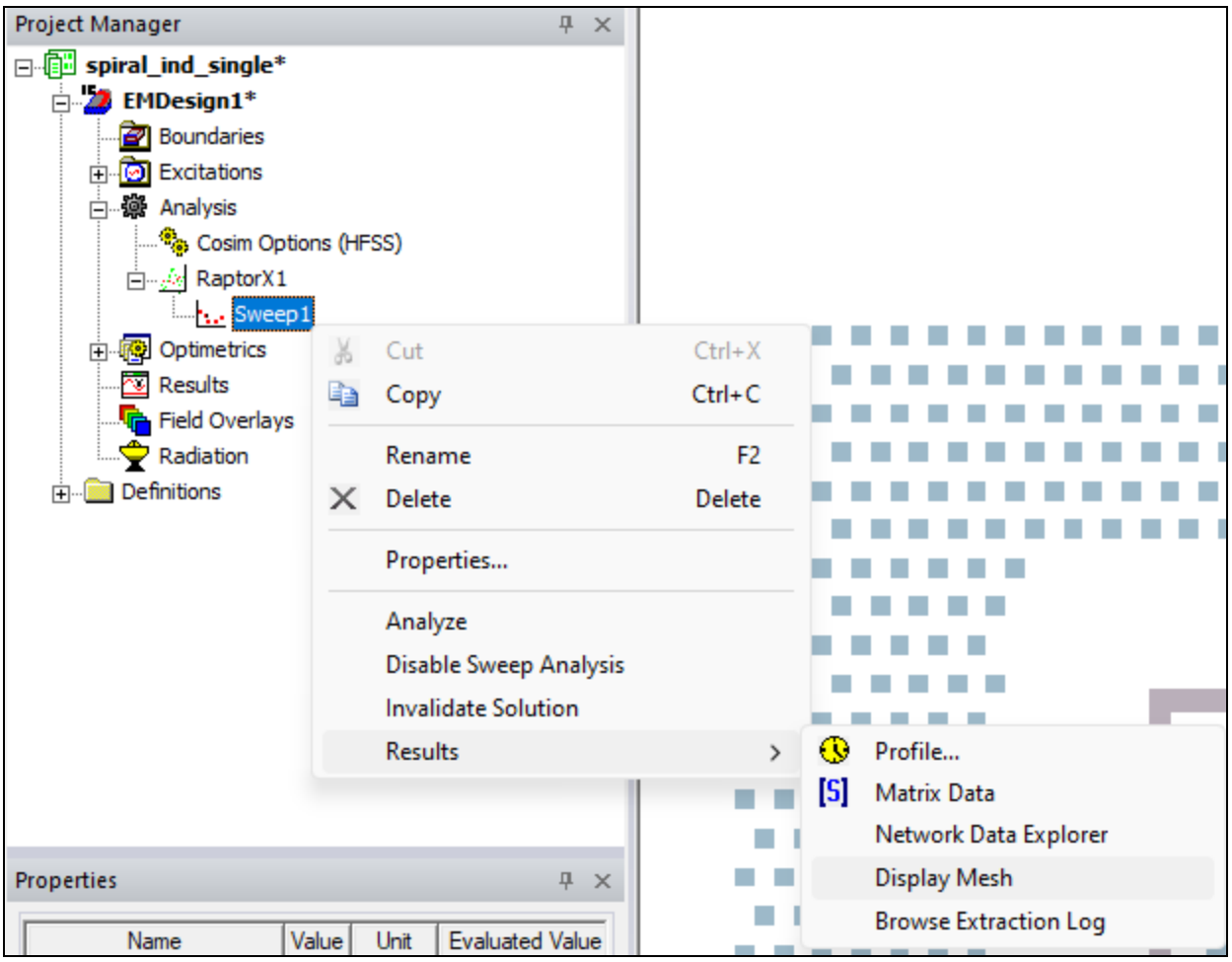


If this is the first time the design has been analyzed with a RaptorX solution setup, Green's Function Data is generated during the analysis.



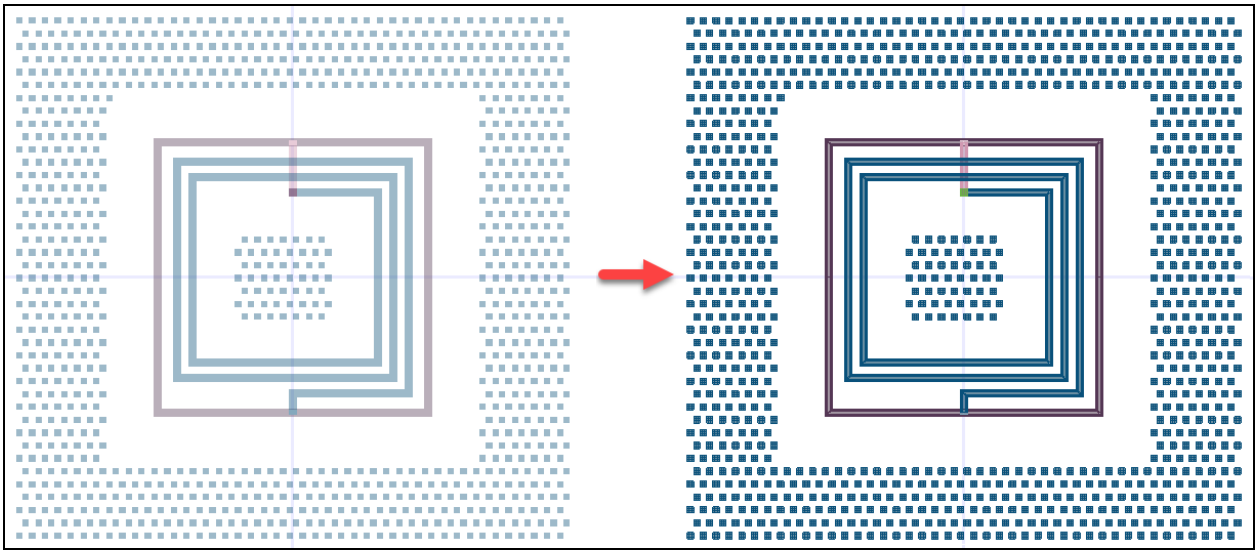
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Once the simulation is complete, a mesh visualization is generated. Navigate to the **Project Manager** window and expand **RaptorX1**. Then right-click **Sweep1** and select **Results** > **Display Mesh**.



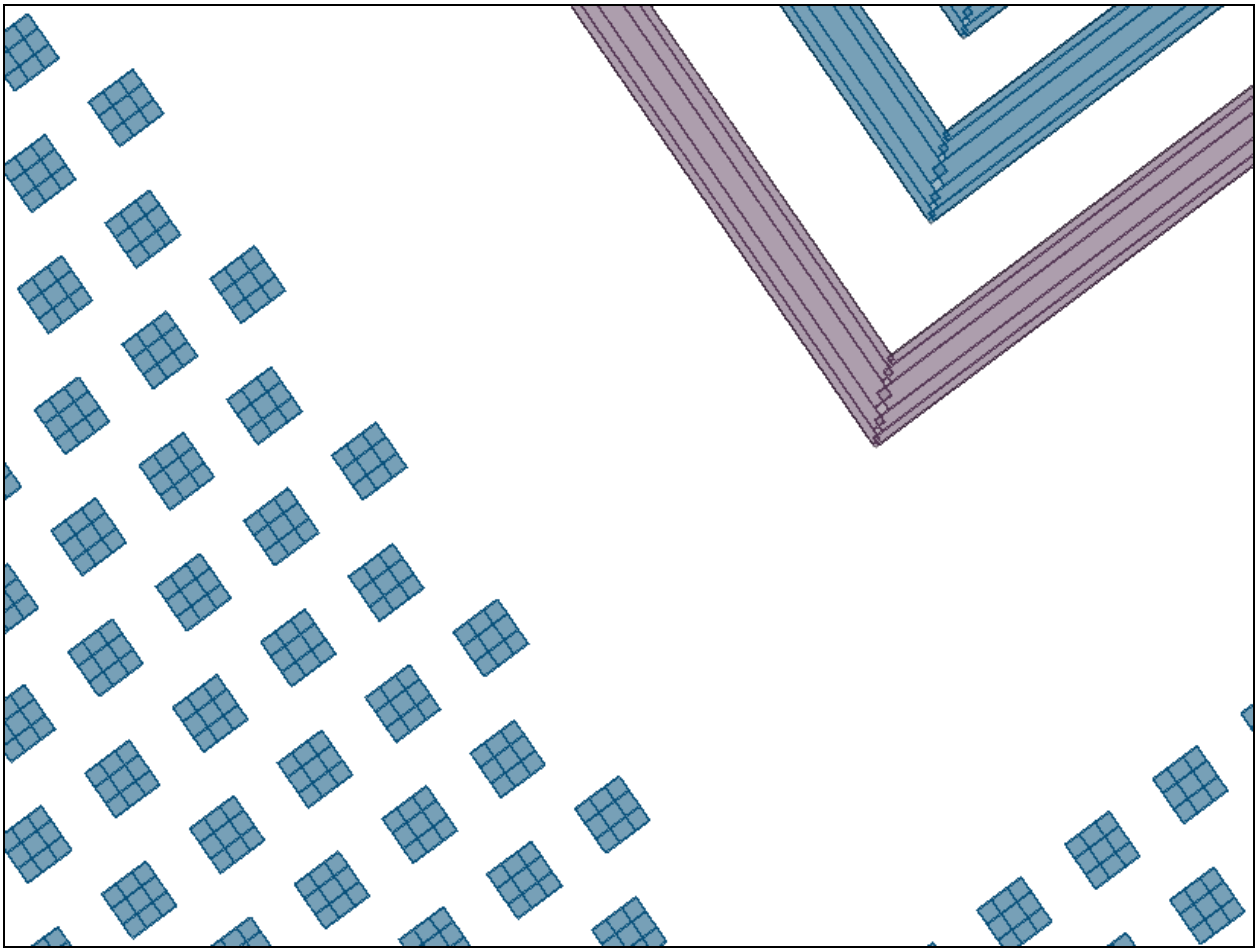
The mesh visualization overlay immediately appears on the design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



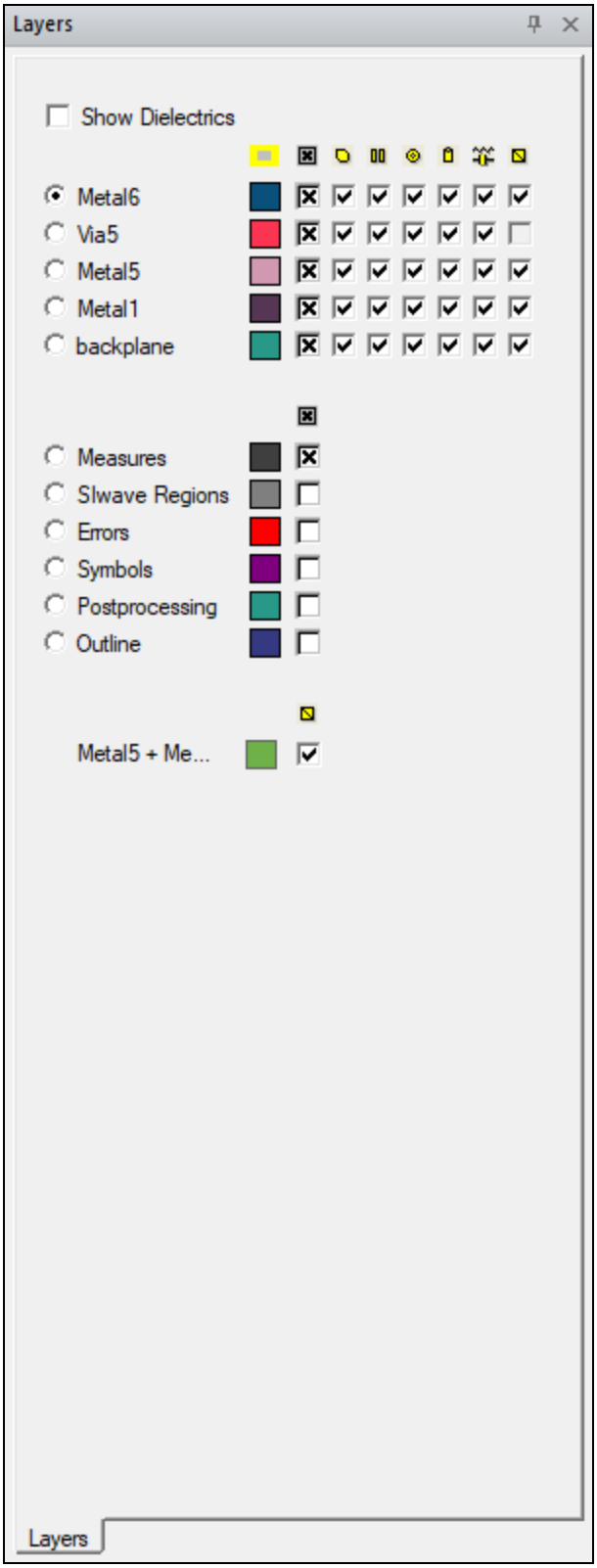
3. Reorient the design to view the mesh visualization from different angles.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. From the **Layers** window, control which layers of the mesh visualization are visible.

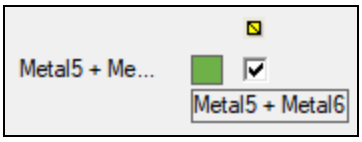
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add or remove checks from the right-hand column (i.e., the **Mesh** column) to show or hide layers of the mesh visualization.

The lowest quadrant of the **Layers** window displays mesh groups created by RaptorX. Refer to **Viewing Green's Function Data**.



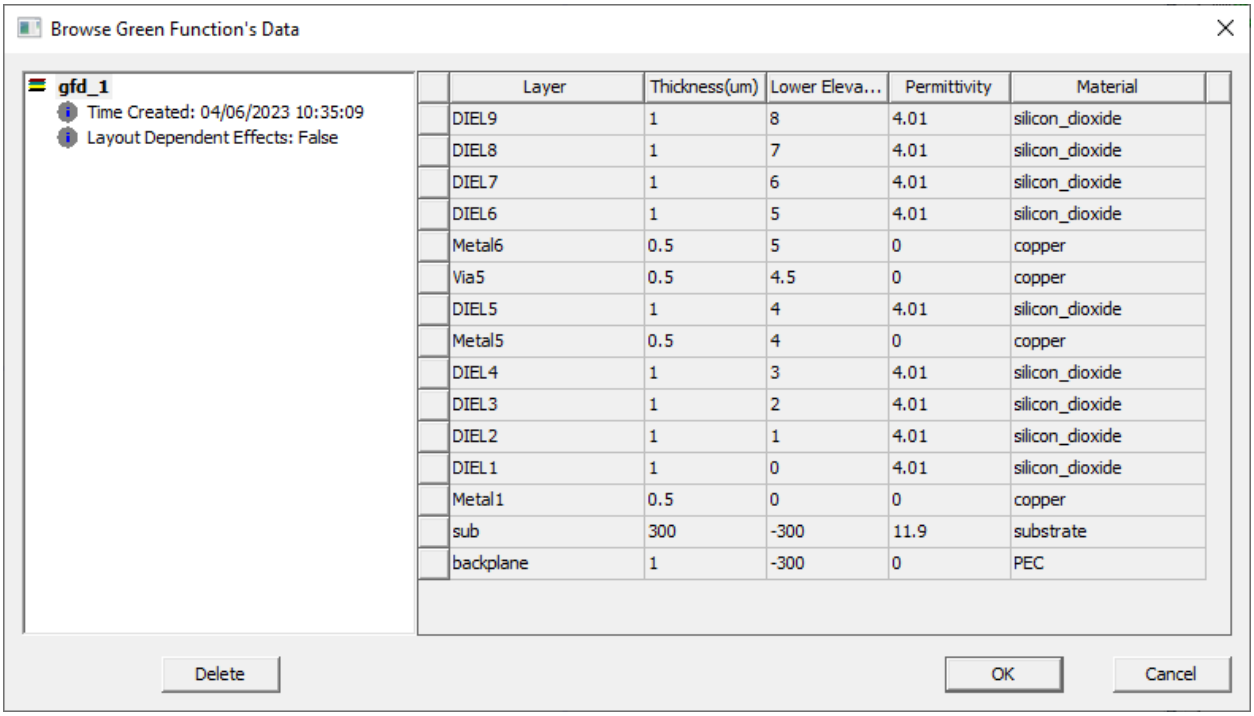
The procedure is complete.

> **Note:**
>
> Certain RaptorX simulations can be distributed over multiple nodes using MPI. RaptorX HPC-integration only works with **Intel MPI** and currently, only auto-distribution setup is supported. All setups will be treated as auto distribution setups by the solver.
>
> Familiarize yourself with the chapter titled, *"Installing MPI for Ansys Parallel Processing"* in the Windows Installation Guide. At the end of the Windows and Linux installation processes, links are provided to access the Intel MPI installation program and its documentation. Install and configure MPI before attempting to run Distributed solutions.

Viewing Green's Function Data

Either during analysis of a RaptorX solution setup or by selecting **Generate Green's Function Data**, the design's current stackup parameters are compiled and entered into a reference database. If the design is not altered, reusing the gathered data will speed up future RaptorX simulations.

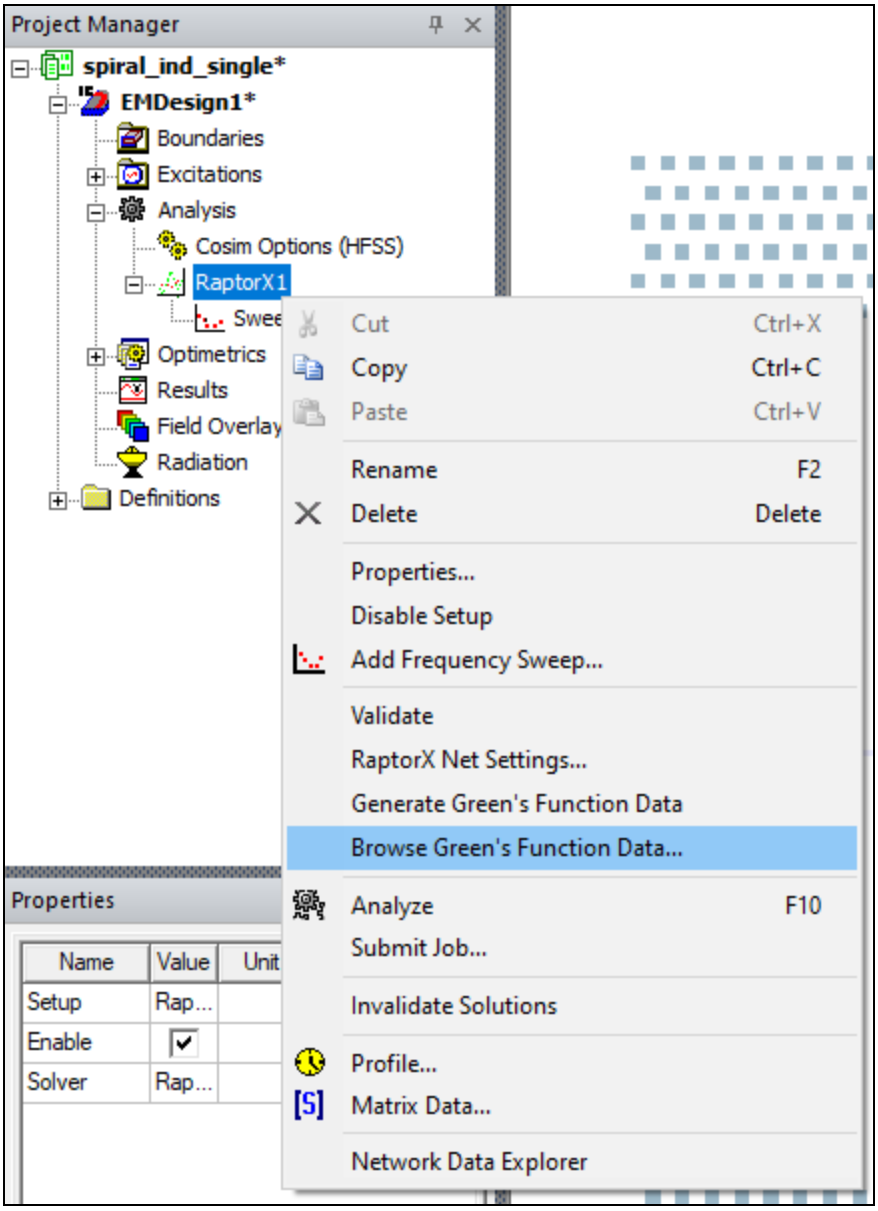
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To open the **Browse Green's Function Data** window, do either of the following:

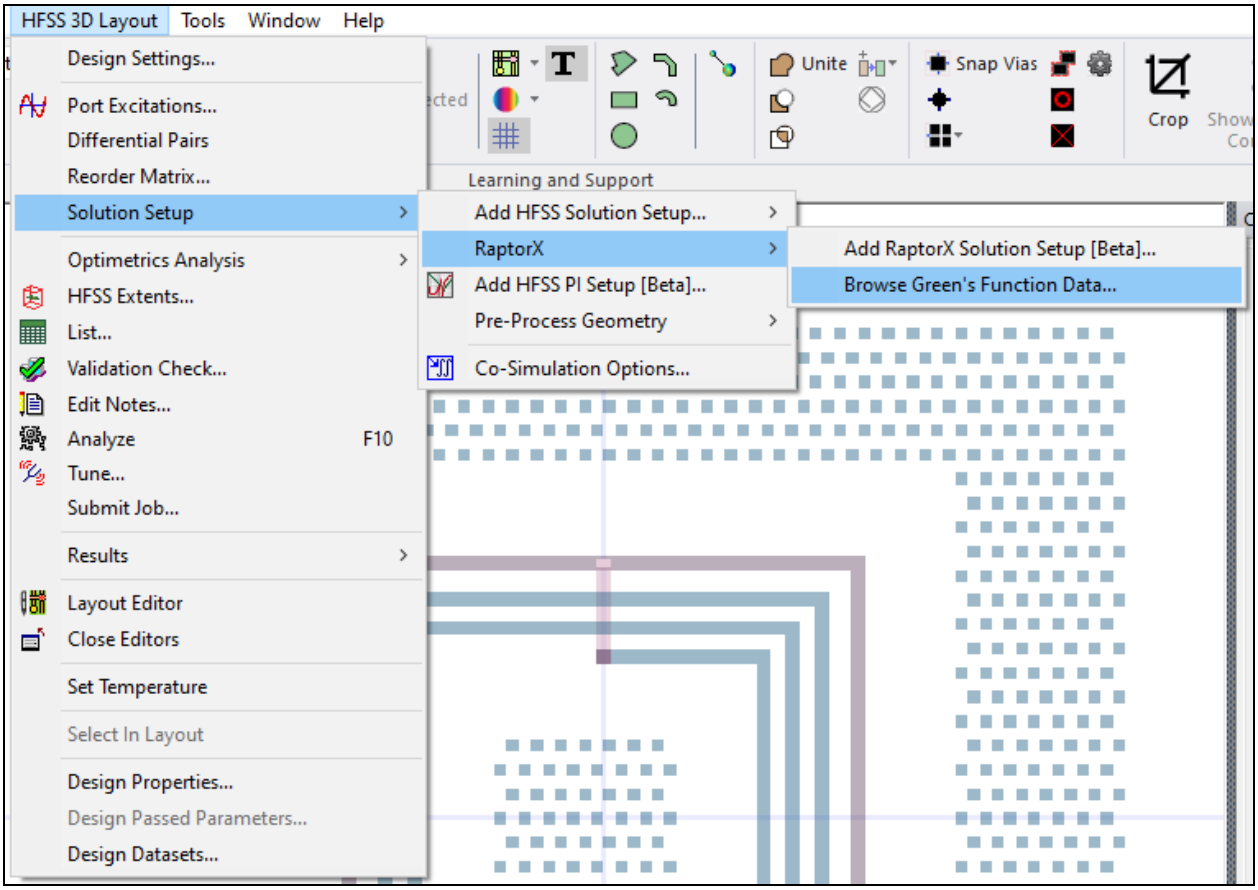
- From the **Project Manager** window, right-click the RaptorX solution setup (e.g., **RaptorX1**) and select **Browse Green's Function Data**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- From **HFSS 3D Layout**, select **Solution Setup** > **RaptorX** > **Browse Green's Function Data**.

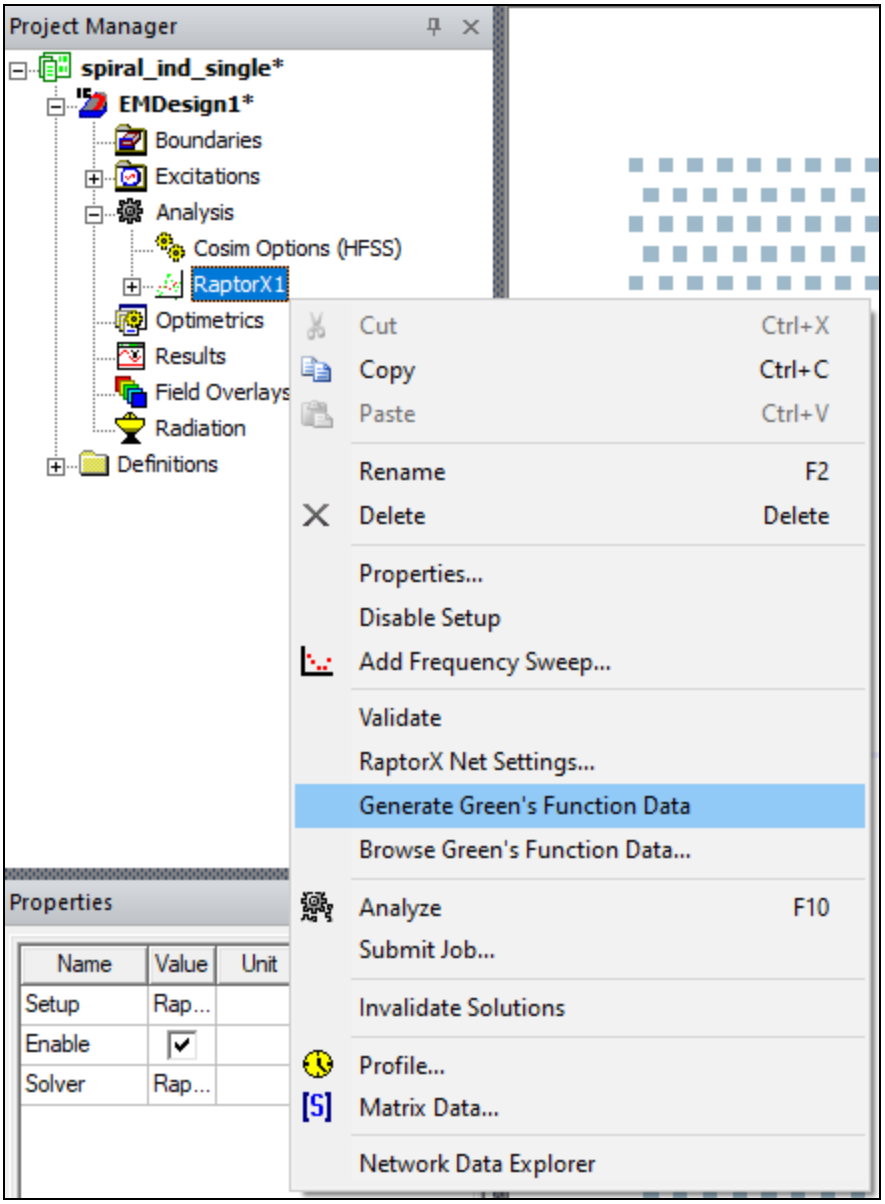
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Generating Green's Function Data

To generate Green's function data before running a simulation, navigate to the **Project Manager** window. Right-click the RaptorX solution setup (e.g., **RaptorX1**) and select **Generate Green's Function Data**.

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 Q3D Solution Setup

The Q3D solution setup mirrors the capabilities of a simulation run using Q3D Extractor. Follow these steps to add a Q3D solution setup to a project.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

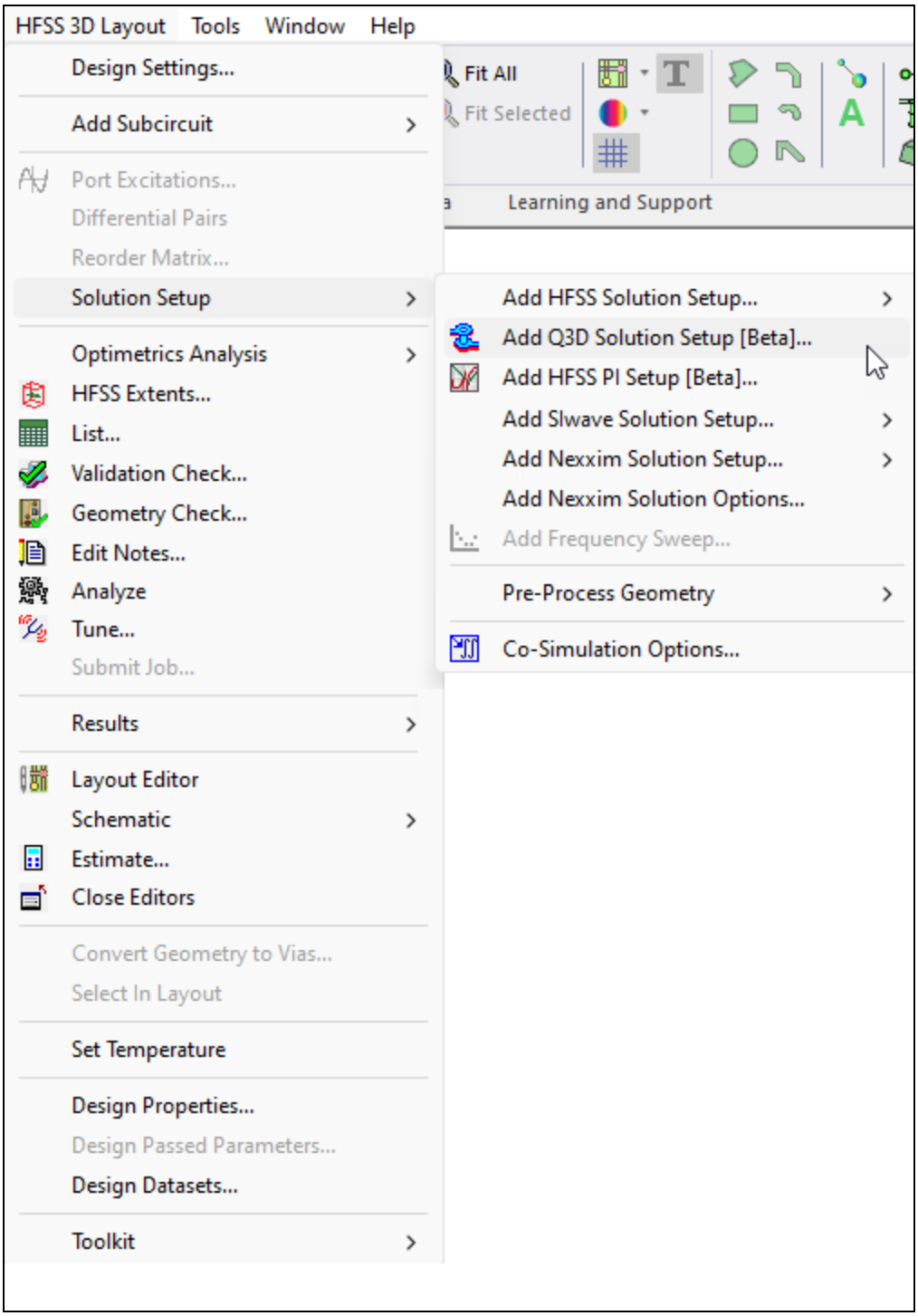
The Q3D solution setup is a beta feature in General mode. Beta features must be activated prior to use. Refer to Activating the Q3D Solution Setup Beta Feature for instructions.

1. Open the **Q3D Solution Setup** window by doing one of the following:
   - From the **Project Manager** window, expand the **Project Tree** > [*Active Design Folder*] (e.g., **Example Project** > **Example Design**). Then right-click **Analysis** and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

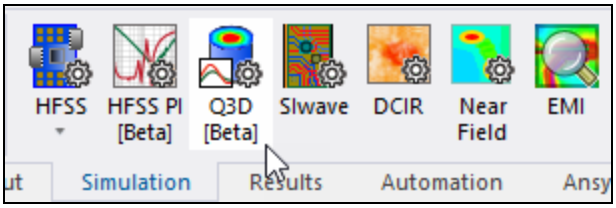
select **Add Q3D Solution Setup**.

- From **HFSS 3D Layout**, select **Solution Setup** > **Add Q3D Solution Setup**.



- From the **Simulation** tab, select **Q3D**.

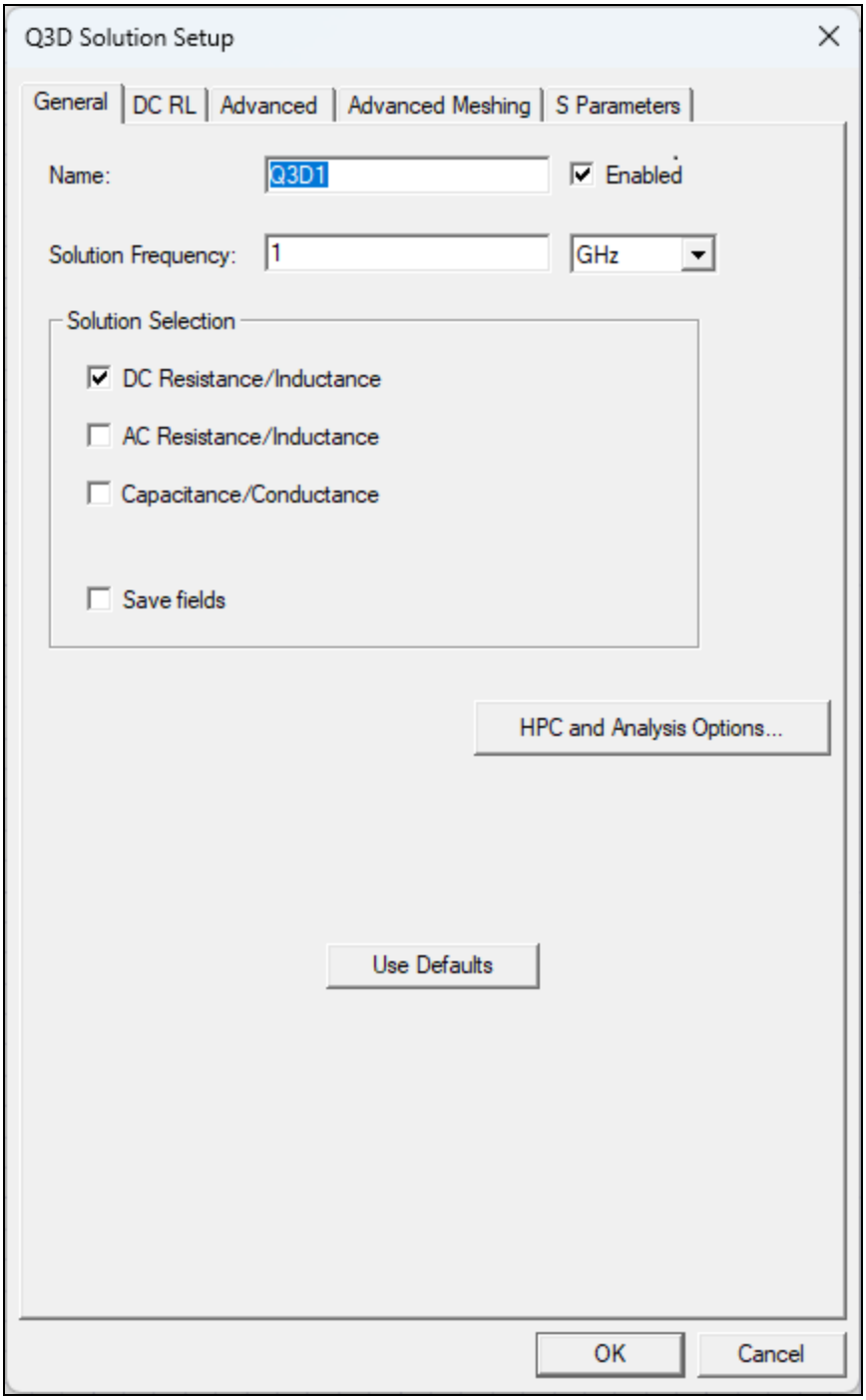
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Q3D Solution Setup** window, make selections from the visible tabs (i.e., by default, **General**, **DC RL**, **Advanced**, **Advanced Meshing**, and **S Parameters** tabs are visible). Default values shown.

> **Note:**
>
> The **Advanced Meshing** tab only appears in **General** mode.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



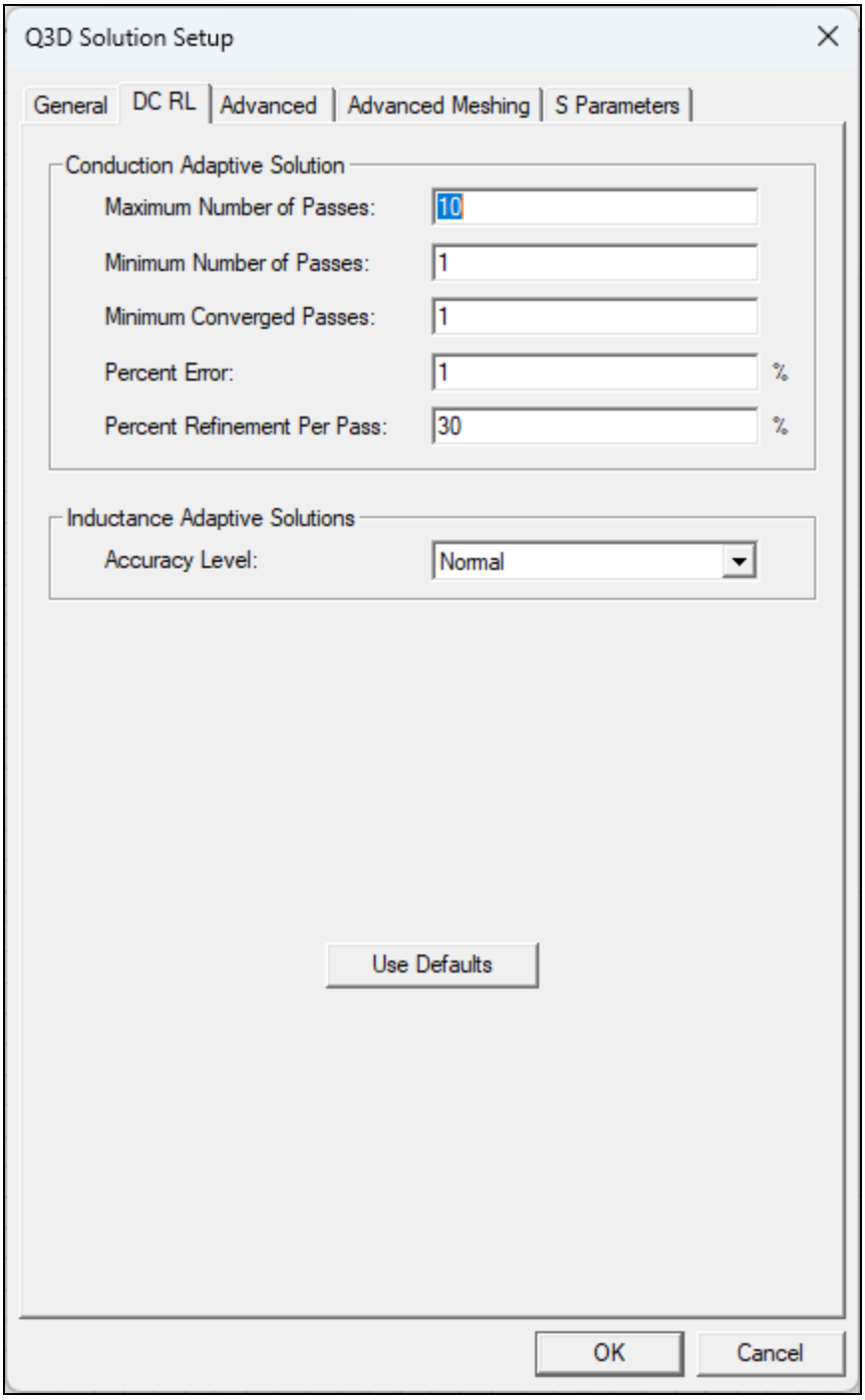
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The **CG** and **AC RL** tabs appear depending on selections made in the **General** tab; **DC RL** is visible by default, but is likewise hidden if the check is removed from the **DC Resistance/Inductance** box).

3. From the **General** tab, set the following:
   - **Name** – type a name in the field or accept the default.
   - **Enabled** – check the box to activate the setup.
   - **Solution Frequency** – enter a value in the provided field to specify the simulation frequency and select a unit of measurement from the corresponding drop-down menu.
   - **DC Resistance/Inductance** – checked by default to solve and generate mesh for for DC resistance and inductance; activates the **DC RL** tab.
   - **AC Resistance/Inductance** – check the box to solve and generate mesh for AC resistance and inductance; activates the **AC RL** tab.
   - **Capacitance/Conductance** – check the box to solve and generate mesh for DC capacitance and conductance; activates the **CG** tab.
   - **Save fields** – check the box to save the current settings as the new default settings.
   - **HPC and Analysis Options** – click to open the **HPC and Analysis Options** window. Refer to Setting 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.

4. Click the **DC RL** tab.



5. From the **DC RL** tab, set the following DC resistance/inductance options:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Maximum Number of Passes** – enter a value in the provided field to specify the maximum number of mesh refinement cycles to perform. This value is a stopping criterion for the adaptive solution; the simulation continues until the maximum number of passes complete or convergence criteria are reached.

  > **Important:**
  >
  > The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform requesting more memory than available or taking excessive time to compute solutions.

- **Minimum Number of Passes** – enter a value in the provided field to specify the minimum number of mesh refinement cycles. Simulation continues until *after* this number of passes complete.

  Minimum Converged Passes – enter a value in the provided field to specify the minimum number of passes that must meet convergence criteria before the simulation stops.

- **Percent Error** – enter a value in the provided field to specify an appropriate solution accuracy. Larger values produce solutions faster, but with less accurate results.

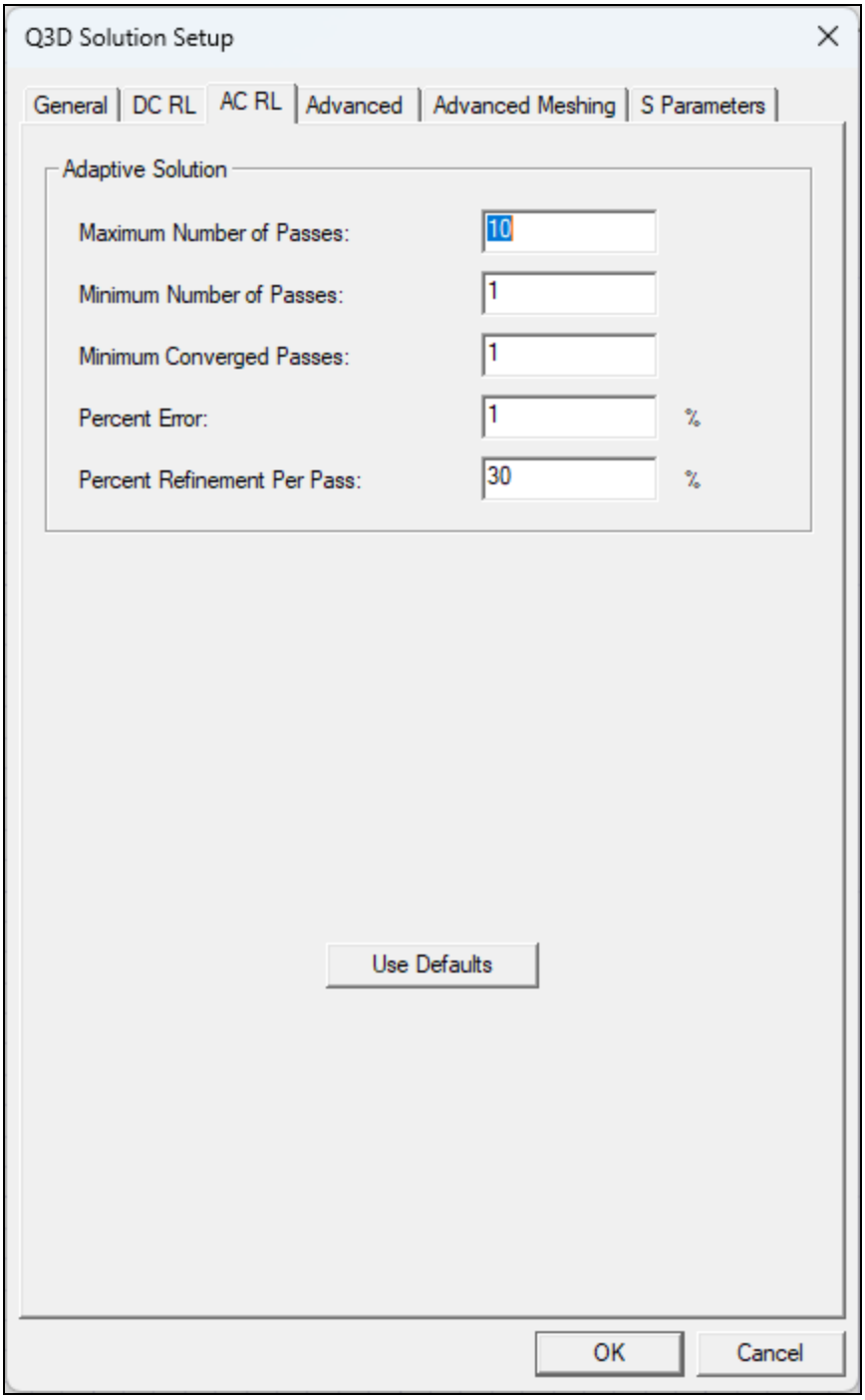
- **Percent Refinement Per Pass** – enter a value in the provided field to specify how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error are refined (e.g., entering 10 causes the mesh to increase approximately 10% each pass. If the mesh consisted of 1000 elements, the tetrahedra are refined such that 100 new elements are added to the mesh). The default percentage works well for most applications.

  > **Note:**
  >
  > A growth rate lower than the maximum occurs when there are not enough high error elements to warrant adaption. This reduces memory usage without sacrificing accuracy.

- **Accuracy Level** – select an inductance adaptive solution accuracy from the drop-down menu. The solver will give accurate results at the **Normal** setting for most applications. The **High**, **Higher**, and **Highest** options offer greater accuracy at the expense of speed and memory.

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 visible, click the **AC RL** tab.



7. From the **AC RL** tab, set the following AC resistance/inductance options:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Maximum Number of Passes** – enter a value in the provided field to specify the maximum number of mesh refinement cycles to perform. This value is a stopping criterion for the adaptive solution; the simulation continues until the maximum number of passes complete or convergence criteria are reached.

  > **Important:**
  >
  > The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in AEDT requesting more memory than available or taking excessive time to compute solutions.

- **Minimum Number of Passes** – enter a value in the provided field to specify the minimum number of mesh refinement cycles. Simulation continues until *after* this number of passes complete.

  Minimum Converged Passes – enter a value in the provided field to specify the minimum number of passes that must meet convergence criteria before the simulation stops.

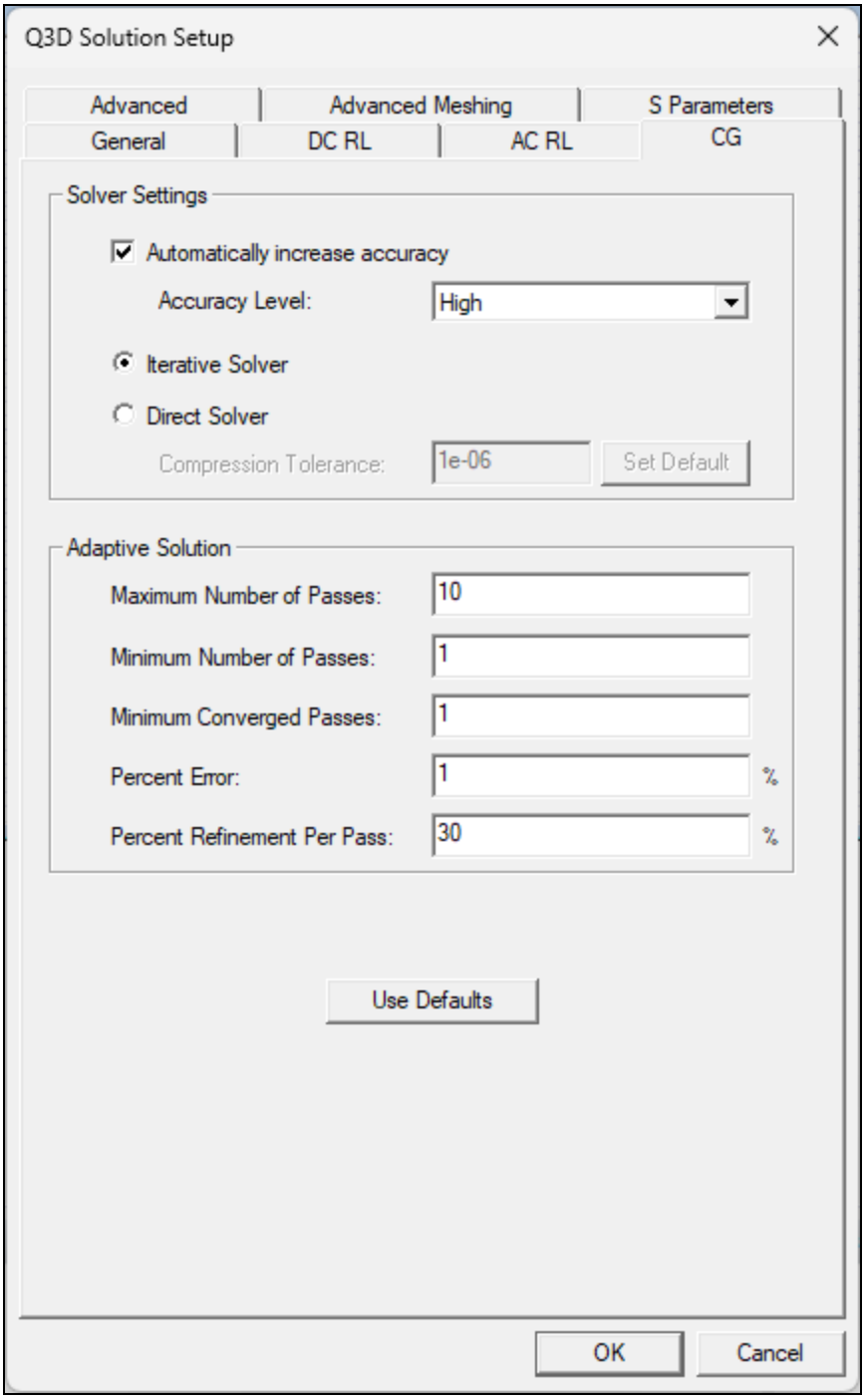
- **Percent Error** – enter a value in the provided field to specify an appropriate solution accuracy. Larger values produce solutions faster, but with less accurate results.

- **Percent Refinement Per Pass** – enter a value in the provided field to specify how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error are refined (e.g., entering 10 causes the mesh to increase approximately 10% each pass. If the mesh consisted of 1000 elements, the tetrahedra are refined such that 100 new elements are added to the mesh). The default percentage works well for most applications.

  > **Note:**
  >
  > A growth rate lower than the maximum occurs when there are not enough high error elements to warrant adaption. This reduces memory usage without sacrificing accuracy.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. If visible, click the **CG** tab.



9. From the **CG** tab, set the following DC capacitance/conductance options:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Automatically increase accuracy** – check the box to automatically increase accuracy, then select an **Accuracy Level** from the drop-down menu. The solver will give accurate results at the **Normal** setting for most applications. The **High**, **Higher**, and **Highest** options offer greater accuracy at the expense of speed and memory.
- Select either of the following:
    - **Iterative Solver** – check the box to activate the **Iterative Solver**, which is generally the fastest option. However, the **Iterative Solver** uses FMM compression, which may lead to slow convergence of GMRES iterations for poorly conditioned matrices.
    - **Direct Solver** – check the box to activate the **Direct Solver**, which has a higher setup time and uses more memory than the **Iterative Solver**. During setup, the **Direct Solver** performs an LU factorization of the FMM-compressed MoM system of equation. From the solution phase, each right hand side is solved using forward(L) and backward(U) substitutions. The **Direct Solver** converges quickly for poorly conditioned matrices and should only be used in cases requiring longer solution times (i.e., cases with extreme geometries and cases with more than 500 nets). After selection, enter a **Compression Tolerance** value. By default, the tolerance is ***1e-6*** for **Normal** or **High** accuracy, ***1e-7*** for **Higher** accuracy, and ***ie-8*** for **Highest** accuracy. Setting a larger tolerance requires less setup time and memory, while setting a tighter tolerance requires more setup time and memory but leads to rapid convergence and faster solution time.
- **Maximum Number of Passes** – enter a value in the provided field to specify the maximum number of mesh refinement cycles to perform. This value is a stopping criterion for the adaptive solution; the simulation continues until the maximum number of passes complete or convergence criteria are reached.

> **Important:**
>
> The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in AEDT requesting more memory than available or taking excessive time to compute solutions.

- **Minimum Number of Passes** – enter a value in the provided field to specify the minimum number of mesh refinement cycles. Simulation continues until *after* this number of passes complete.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

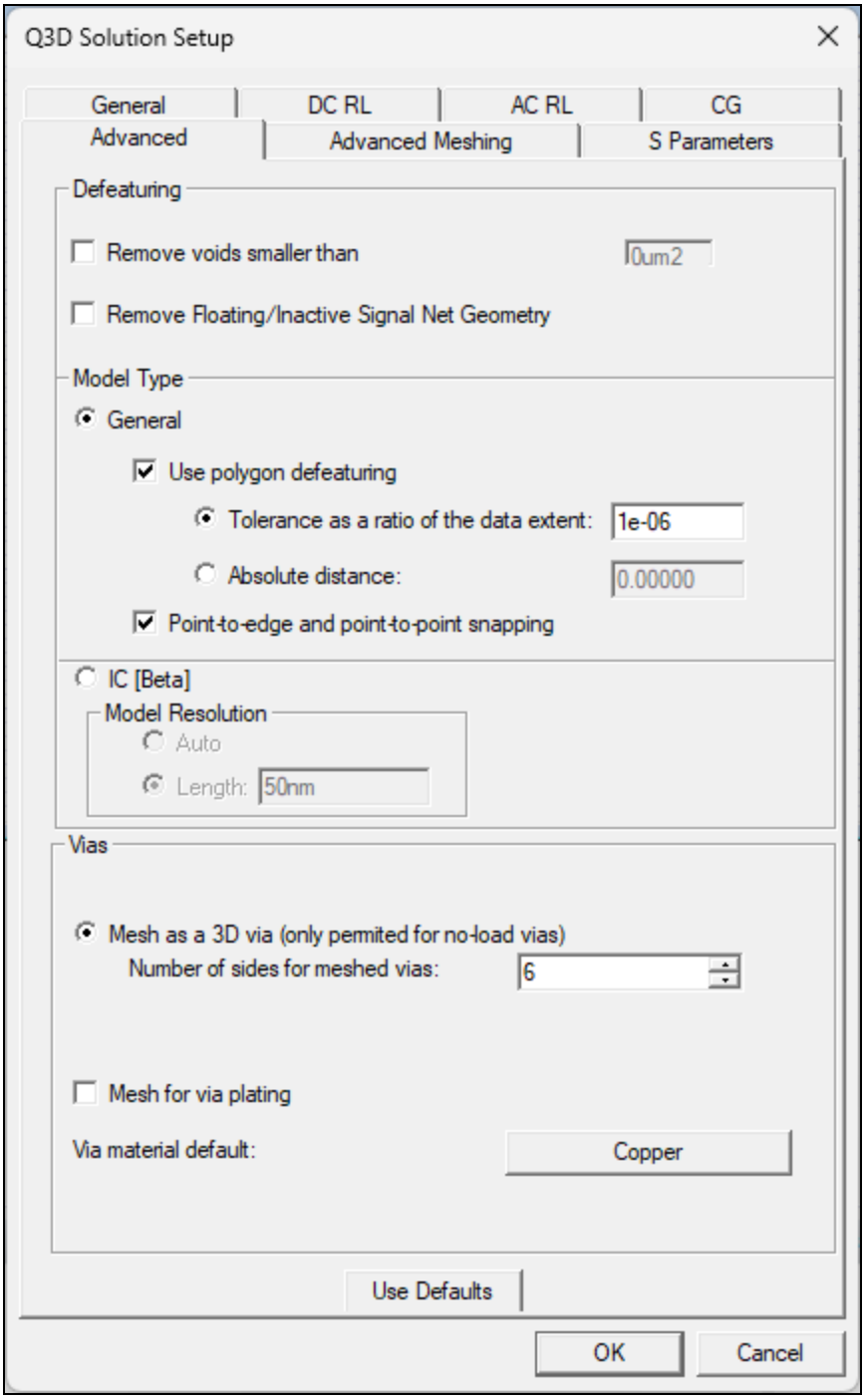
Minimum Converged Passes – enter a value in the provided field to specify the minimum number of passes that must meet convergence criteria before the simulation stops.

- **Percent Error** – enter a value in the provided field to specify an appropriate solution accuracy. Larger values produce solutions faster, but with less accurate results.
- **Percent Refinement Per Pass** – enter a value in the provided field to specify how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error are refined (e.g., entering 10 causes the mesh to increase approximately 10% each pass. If the mesh consisted of 1000 elements, the tetrahedra are refined such that 100 new elements are added to the mesh). The default percentage works well for most applications.

> **Note:**
>
> A growth rate lower than the maximum occurs when there are not enough high error elements to warrant adaption. This reduces memory usage without sacrificing accuracy.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Click the **Advanced** tab.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. From the **Advanced** tab, set the following:

    - **Remove voids with an area smaller than** – check the box to remove voids from the simulated primitives. Enter a tolerance value in the field. Voids smaller than the tolerance value will be removed.
    - **Remove Floating/Inactive Signal Net Geometry** – check the box to remove floating or inactive signal net geometry from the simulation.
    - **Model Type** – select either of the following:
        - **General** offers standard mesh processing.
            - **Use polygon defeaturing** – check the box to remove very close points, points that do not contribute to the geometry of an edge (e.g., collinear), and very thin intrusions. Used with “unioning”, to assist healing geometry that is not snapped together. Then select either of the following:
                - **Tolerance as a ratio of the data exent** – tolerance value specified as a ratio of the overall data extent. This value must be very small.
                - **Absolute distance** – tolerance specified as an absolute value (e.g., 0.0001mm).
            - **Point-to-edge and point-to-point snapping** – check the box to snap points to edges and points to points if they are within a minimum dimensional tolerance based on geometry extents.
        - **IC [Beta]** optimizes mesh processing to simulate on-chip structures. Select either of the following:
            - **Auto** – the mesher automatically calculates **Model Resolution** length based on each object's effective thickness.
            - **Length** – the mesher uses the manually-specified **Model Resolution** length and unit of measurement.

    > **Note:**
    >
    > Vias may be modeled as a simple 'wirebond' or as a 3D ribbon or solid. The values specified in the analysis setup only apply to vias that do not have specific property overrides. A ll properties explicitly specified for a via take precedence over these values.

    - From the **Vias** area, select either of the following:

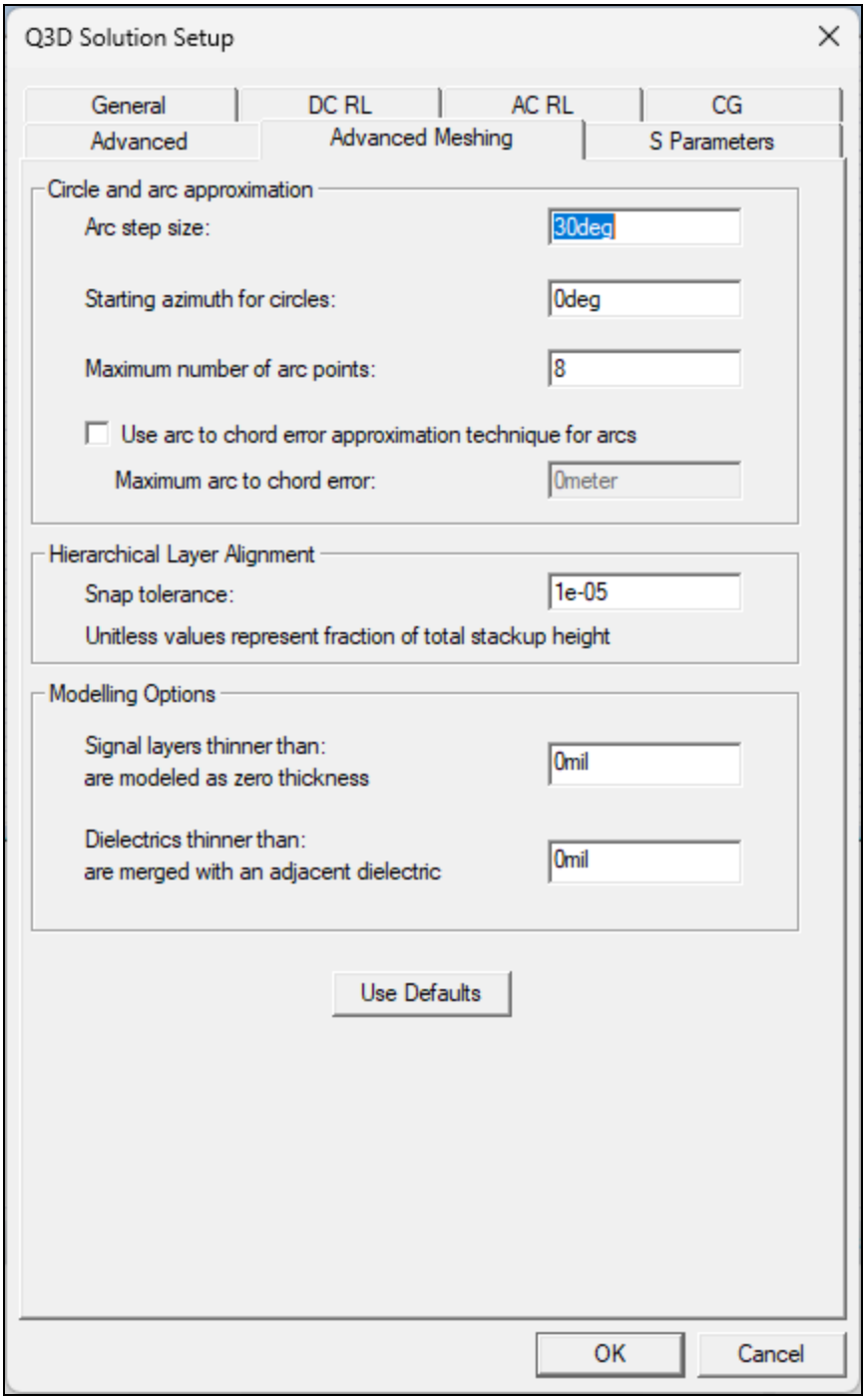
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh as a 3D via** – vias are modeled using rectangles to form a 3D ribbon or solid. Enter a **Number of sides for meshed vias** to use when creating a 3D representation. If the entered value is less than 3, the via is modeled as a flat ribbon.
- **Via Field** – reduces the density of via fields and model them using wirebonds without significant loss in accuracy. A via field is defined by vias that connect two signal layers without intervening geometry) and attach to connected geometry at their upper and lower points. Vias that span multiple layers are broken up vertically at all intersection points. Each segment is treated independently. Two vias may be adjacent and close, but they are not thinned if they connect to electrically separate geometry either at their upper or lower connection points. Doing so would break the electrical connection. Then enter a **Relative minimum via spacing** to define a radius around a via. Any via that falls within the radius is eliminated. The value specified is a multiple of the average via spacing (i.e., the algorithm considers the averages for all via clusters and uses the minimum average spacing. Assuming all vias are spaced equally, a value of 1 eliminates every other via, 2 eliminates 2 vias between each via, et cetera).
- **Mesh for via plating** – check the box to create a mesh around the plating percent specified in the **Edit Padstack Definition** window, resulting in a more accurate DC thickness value. A via that is not fully plated includes a void filled with the material defined in the surrounding air box (e.g., air). Zero plating (i.e., 0%) is not recognized and is treated as fully plated.

- **Via material default** – when a material has not been specified in the padstack definition and there is no override in the properties for a via, then this is the material associated with the via mesh. It is initially set to copper.

12. To continue setup in IC mode, skip to step 14. Otherwise, click the **Advanced Meshing** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



13. From the **Advanced Meshing** tab, set the following:

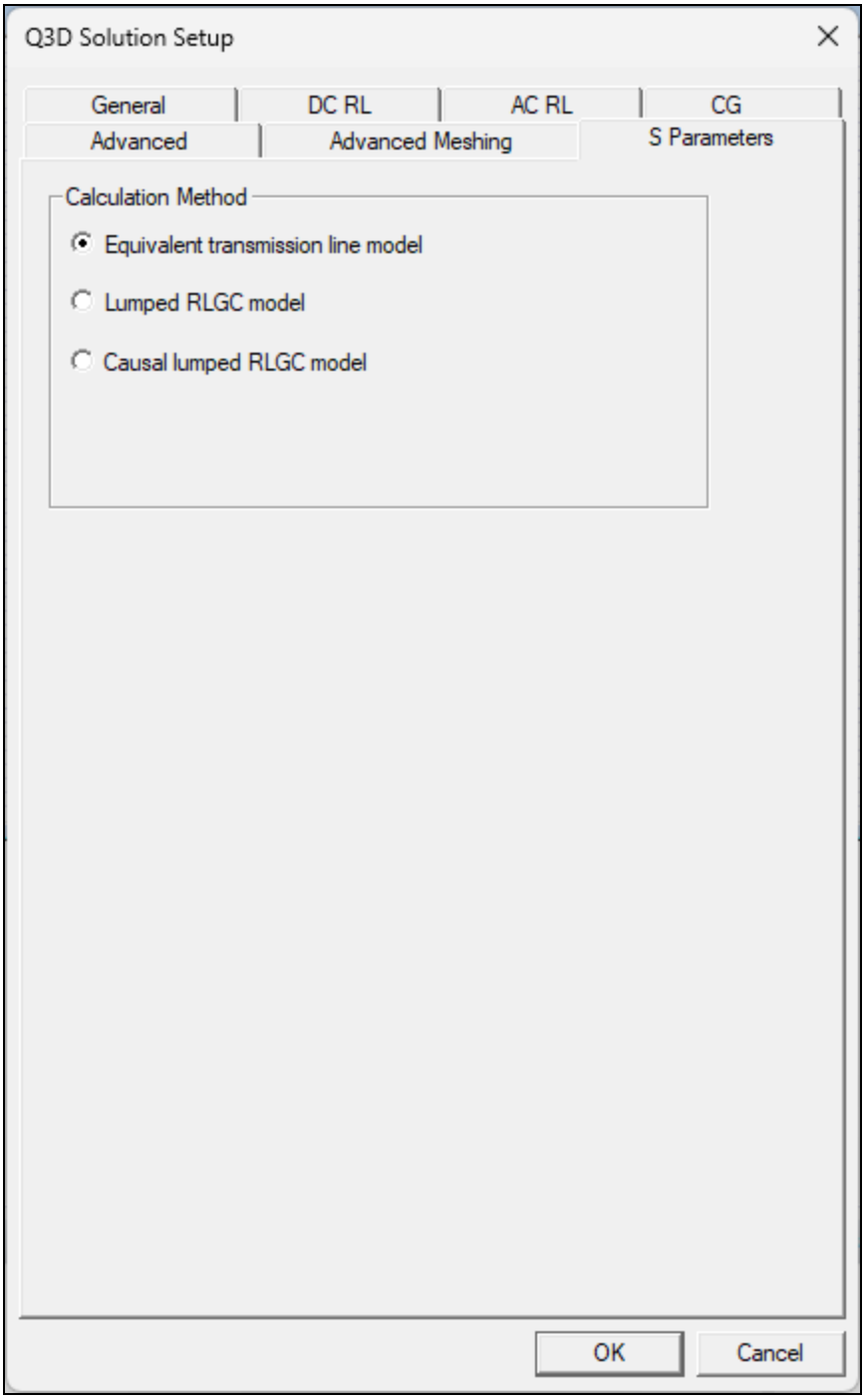
    - **Arc step size** – enter a value to set the angular interval, in degrees, below which mesh points are added.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Starting azimuth for circles** – enter a value to specify the location, in degrees, of the first mesh point or circles.
- **Maximum number of arc points** – enter a value to set the maximum number of mesh points on a given arc segment.
- **Use arc to chord error approximation technique for arcs** – check the box to calculate whether the resulting straight edge approximation deviate from the original arc by more than the specified error. Then enter a **Maximum arc to chord error** value and unit of measurement to set a maximum distance permissible between a straight edge approximation (i.e., chord) and the original arc.
- **Snap Tolerance** – enter a value/variable to set the tolerance by which layers align across hierarchical boundaries.
- **Signal layers thinner than <*value*> are modeled as zero thickness** – enter a value. Negative signal layers in the design which are thinner than the value are modeled as being infinitely thin in order to reduce simulation time.
- **Dielectrics thinner than <*value*> are merged with an adjacent** – enter a value. Dielectric layers in the design which are thinner than the value are merged into an adjacent dielectric. Material properties are accumulated using a weighted average

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. Click the **S Parameters** tab.



15. From the **S Parameters** tab, select one of the following circuit model calculation methods to compute S-parameters from RLGC matrix data:

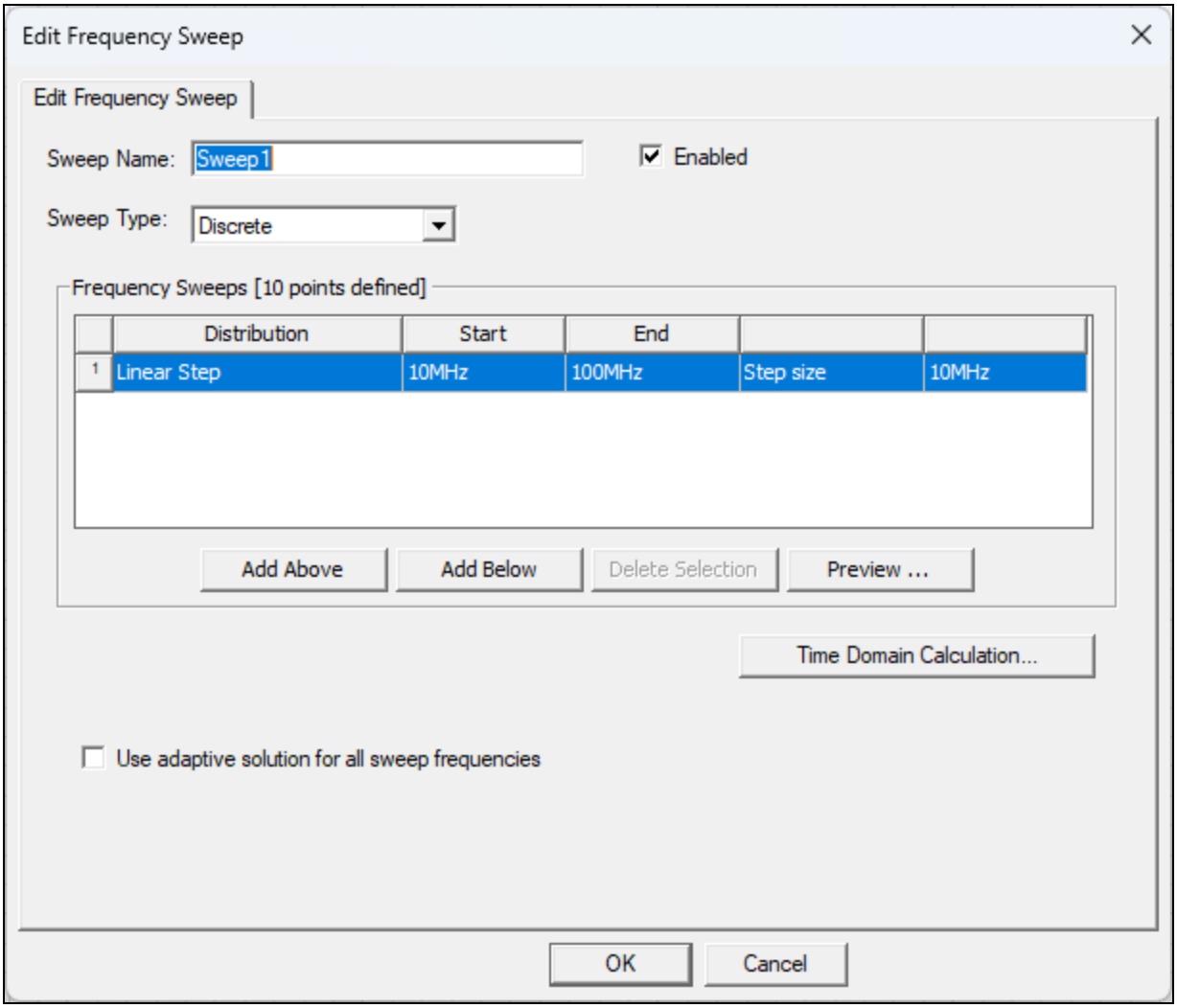
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Equivalent transmission line model** – distributes RLGC parameters across coupled transmission lines. Recommended for designs with parallel interconnects of similar length.
- **Lumped RLGC model** – places inductance and resistance between sources and sinks, and lumps capacitance and conductance at the sink end. Recommended for arbitrary 3D structures.
- **Causal lumped RLGC model** – uses the same topology as the lumped model but incorporates causality via a state-space representation. Recommended for high-frequency designs or co-simulation workflows using Nexxim.

16. Click **OK** to add the Q3D solution setup to the active design and simultaneously open the **Edit Frequency Sweep** window.

> **Note:**
>
> For Q3D solution setups, the frequency sweep is only used to generate S parameters.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



17. From the **Edit Frequency Sweep** window, click **OK** to accept the sweep with its default values or specify the following:

> **Note:**
>
> The available tabs and options change with the selected sweep type.

- **Sweep Name** – enter a name for the sweep.
- **Sweep Type** – select **Discrete** or **Interpolating**:
  - **Discrete** – Generates field solutions at specific frequency points in a frequency range. Ideal when only a few frequency points are appropriate to accurately

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

represent the results in a frequency range.

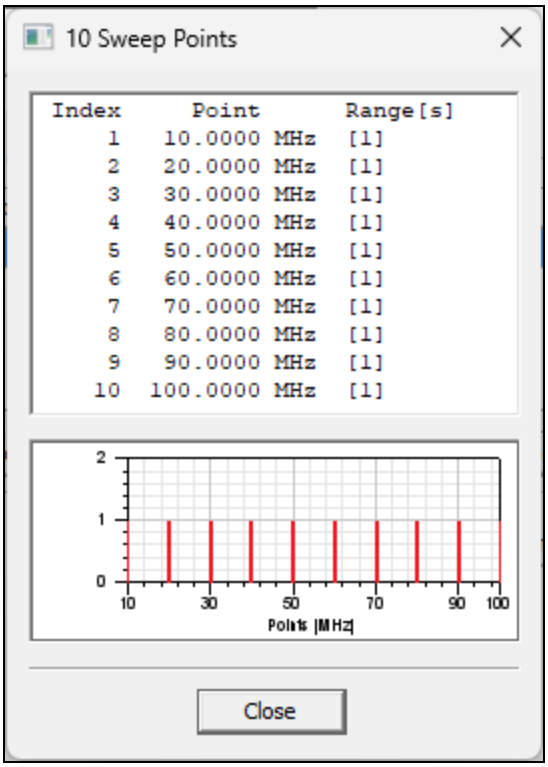
- **Interpolating** – Estimates a solution for an entire frequency range. Ideal when the frequency range is wide and the frequency response is smooth. When Interpolating is selected, the **Interpolation** tab appears.

  > **Note:**
  >
  > The information specified for an Interpolating sweep dictates the amount of information that can be viewed on a post-processing plot.

- **Enabled** – check to enable the sweep.
- **Frequency Sweeps** area – use **Add Above**, **Add Below**, and **Delete** to add or remove sweeps. Each row in the table represents a separate sweep and each column is either a selectable parameter or a value field. If applicable, add multiple frequency sweeps to the same setup. Specify the **Start** and **End** frequency for each sweep, and make selections from the additional options specific to the **Distribution** type.
  - **Distribution** – select from the following:
    - **Linear Step** – A linear range of frequency points for which users specify a constant step size. The simulation will solve the frequency point at each step in the specified frequency range, including the start and stop frequencies (e.g. , specifying 10 for the **Start** frequency, 20 for the **Stop** frequency, and 2.5 for the **Step Size** instructs the simulation to compute a solution for frequencies of 10, 12.5, 15, 17.5, and 20).
    - **Single Point** – Individual frequency points. Insert specific frequency points to solve in the frequency range after uniform frequency points. For **Discrete** sweep types only.
    - **Linear Count** – A linear range of frequency points for which users specify the number (count) of points within the frequency range. The simulation divides the frequency range into the count specified and solve each frequency point in the count.
    - **Log Scale** – A range of points for which users specify **Start**, **End**, and the number of **Samples**.The simulation assigns the sampled points using intervals based on a logarithmic scale.
    - **Use adaptive solution for all sweep frequencies** – check to optimizes the accuracy and efficiency of the solution by refining the mesh or adjusting parameters as needed across all frequencies.

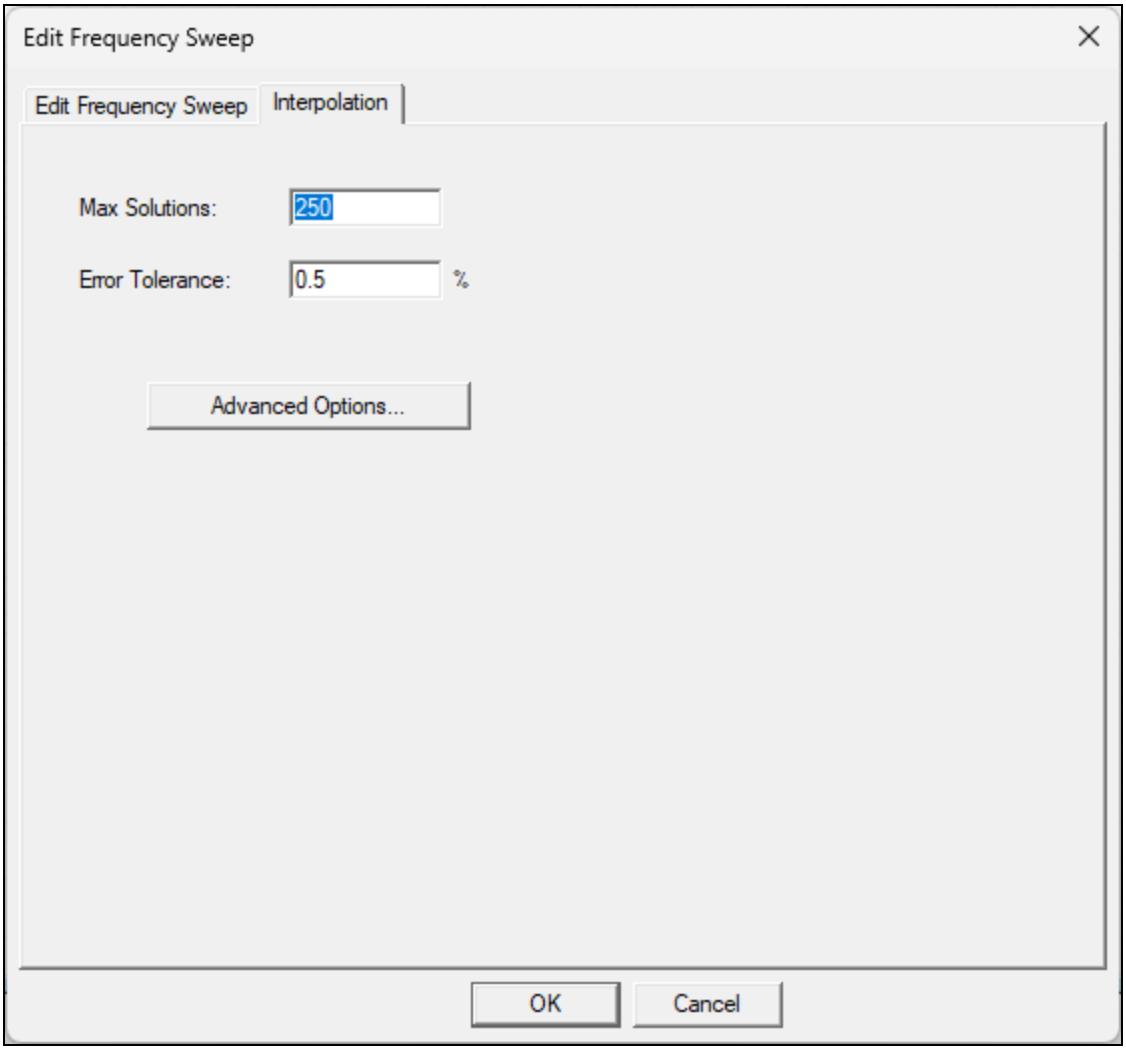
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18. Click **Preview** to view a preview of the sweep, as currently defined.



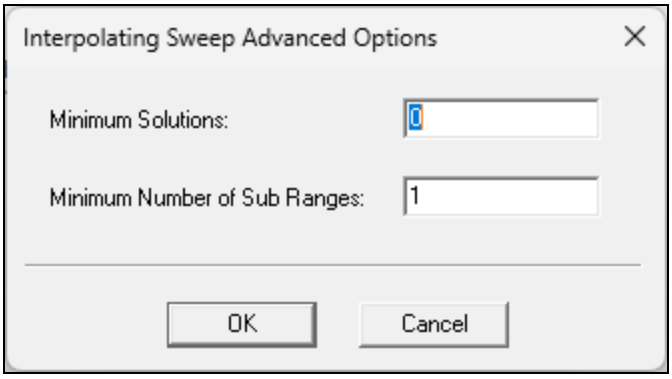
19. If **Interpolating** is selected from the **Sweep Type** drop-down menu, the **Interpolation** tab appears. Click the **Interpolation** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



20. From the **Interpolation** tab, specify the following:
    - **Max Solutions** – the maximum number of solutions that will be solved for the frequency range. The default is **250**.
    - **Error Tolerance** – the maximum relative difference allowed between two successive interpolation solutions. The default is **0.5** percent.
21. Click **Advanced Options** to open the **Interpolating Sweep Advanced 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.

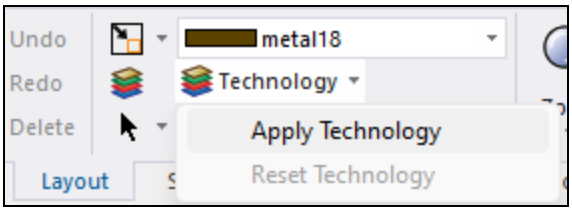


22. From the **Interpolating Sweep Advanced Options** window, specify the following:
    - **Minimum Solutions** – the minimum number of converged solutions that will be solved for the frequency range (e.g., if this value is three, then once the sweep reaches convergence it simulates at two extra frequencies, resembling the minimum number of converged adaptive passes in a regular simulation). Setting a minimum number of solutions can eliminate non-physical S-parameter spikes. The default value is **0**.
    - **Minimum Number of Sub Ranges** – the sub range number acts as an initial condition on the sweep to force initial even breakup of the null range into subranges. The end points and middle of each subrange will be solved, controlling the points at which the interpolating sweep is broken up and preventing redundant effort caused by neighboring interpolating sweeps solving the same point (e.g., 1 GHz-4 GHz and 4 GHz-9 GHz sweeps do not both solve the 4 GHz datapoint). The default value is **1**.
23. Click **OK** to close the **Interpolating Sweep Advanced Options** window.
24. Click **OK** to close the **Edit Frequency Sweep** window and add any sweeps.
25. If any technology files were imported during the initial setup of the design, follow these steps to apply a technology before simulation.

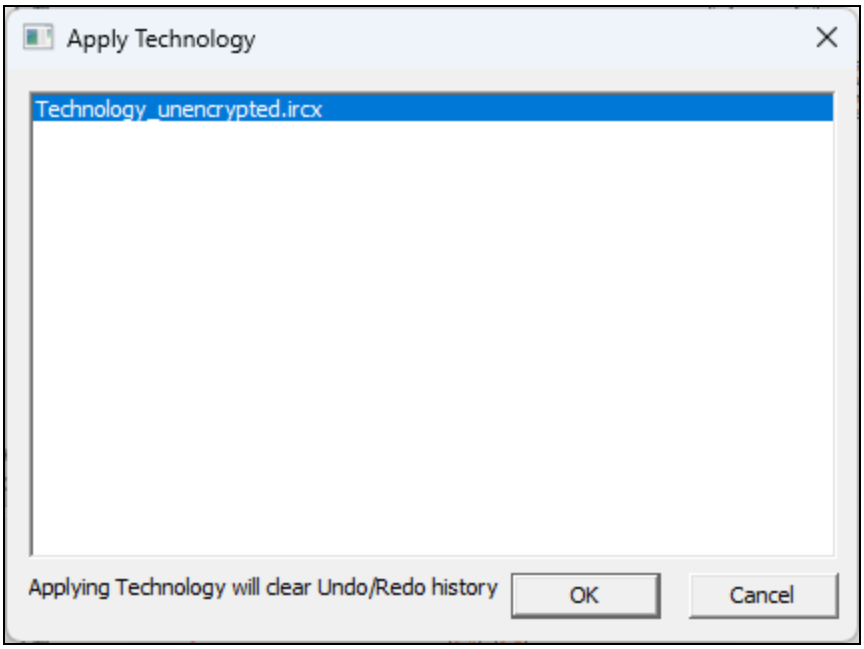
> **Note:**
>
> If technology files were **not** imported during the initial setup of the design, users can add one or more files at any time before simulation. If a technology file **was** applied during setup, the technology's name should already appear in the **Layout** ribbon adjacent to the **Technology** drop-down menu. Users will not need to follow these steps. Refer to Adding a Technology File to a Design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

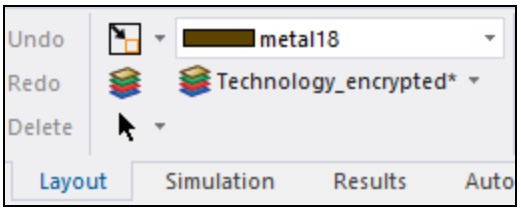
a. If a technology is available, navigate to the **Layout** ribbon and select **Apply Technology** from the **Technology** drop-down menu.



b. Once the **Apply Technology** window opens, select an appropriate technology from the list and click **OK**.

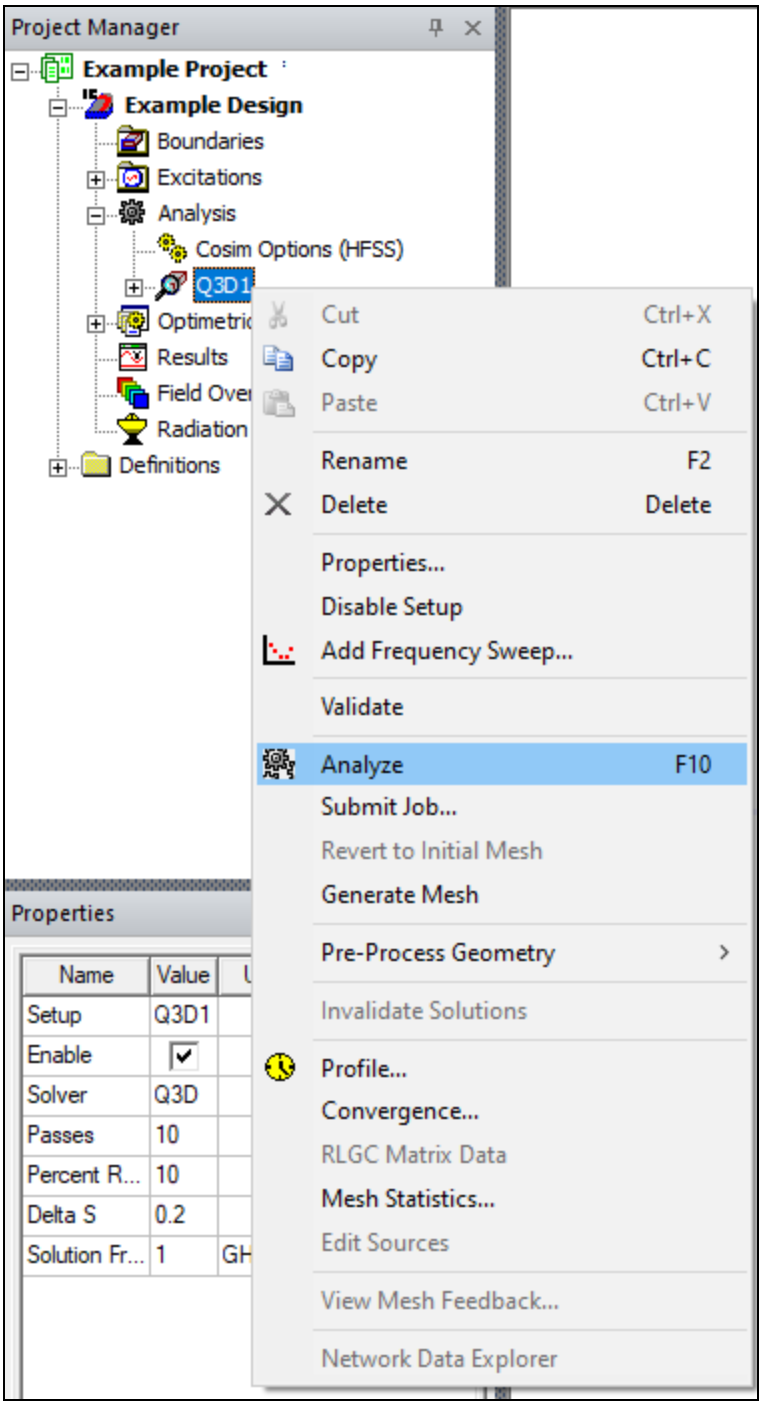


Once the technology is applied to the design, the technology's name appears adjacent to the **Technology** drop-down menu.



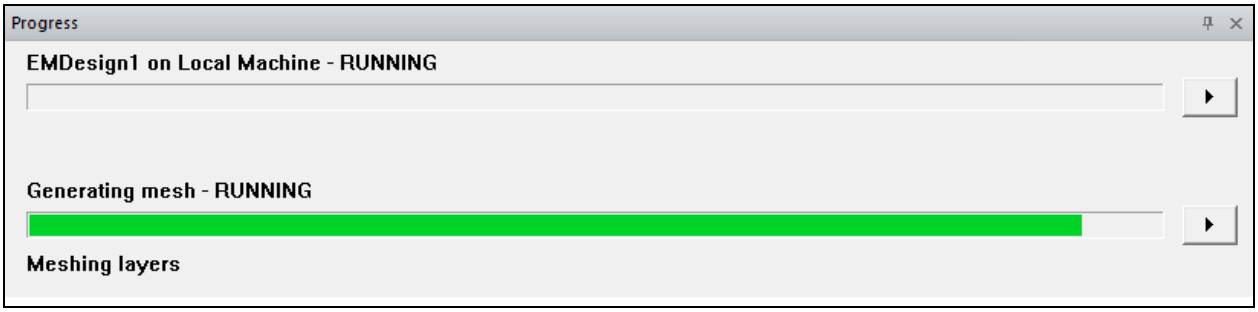
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26. From the **Project Manager** window, right-click the Q3D solution setup (e.g., **Q3D1**) and select **Analyze** to run the simulation. Refer to Running Simulations.

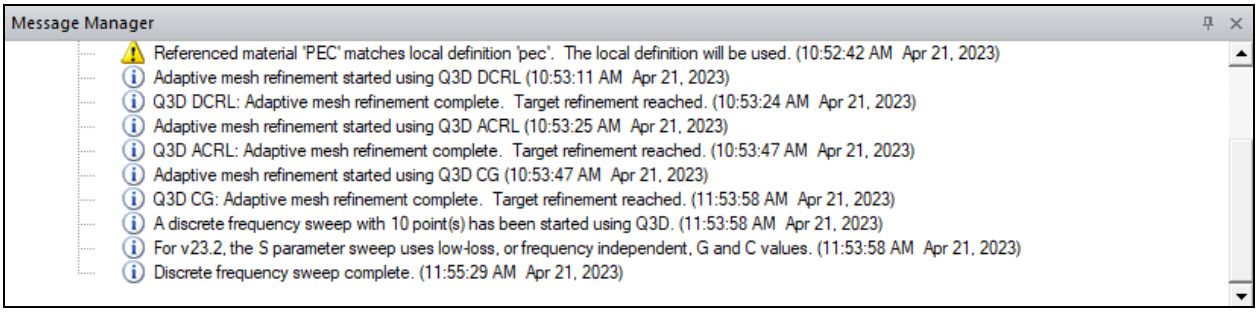


AEDT will begin simulation and generate an appropriate mesh. View, pause, or stop the simulation from the **Progress** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

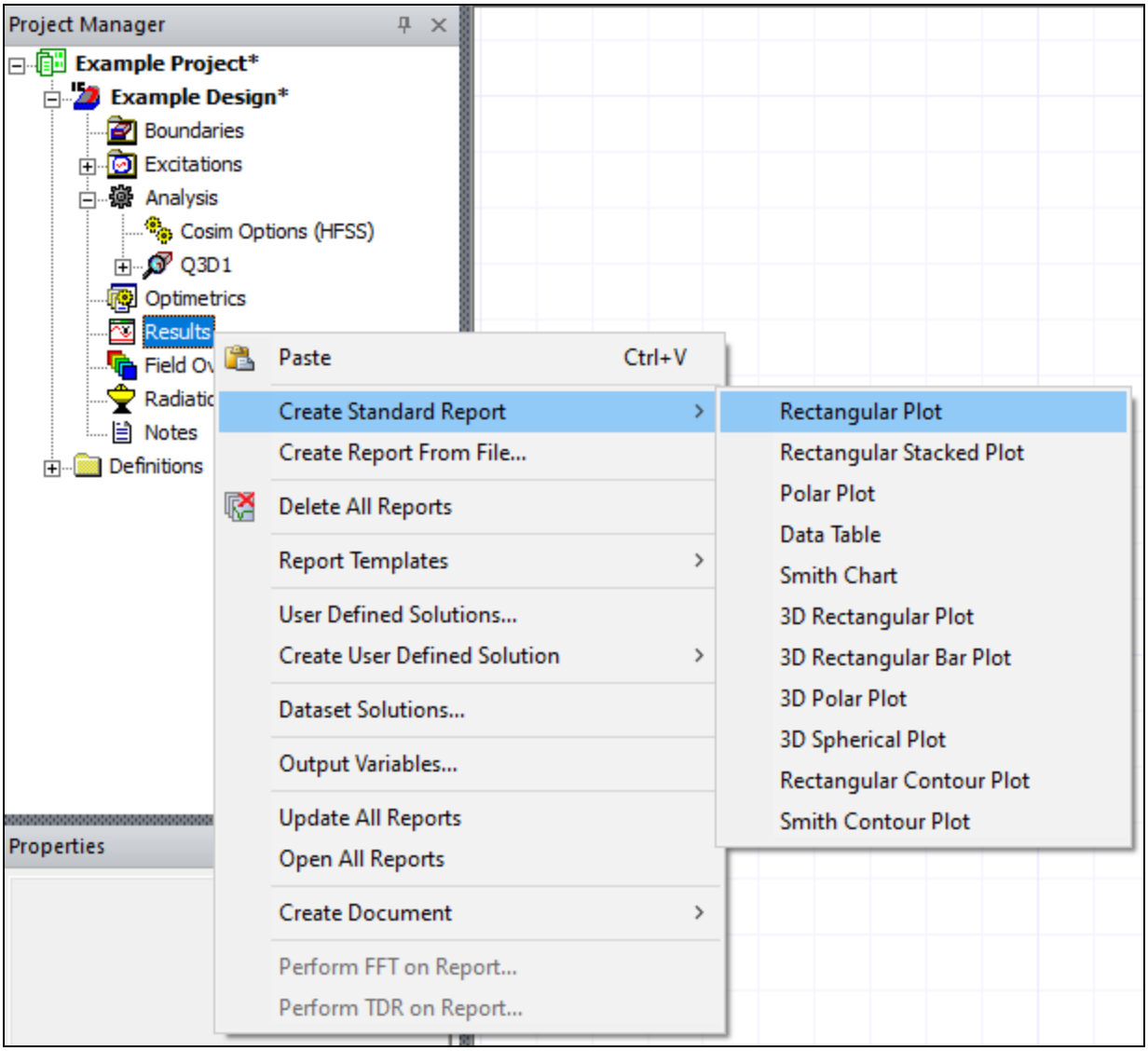


The **Message Manager** window will display pertinent information as the simulation progresses.



27. Once the simulation is complete, navigate to the **Project Manager** window and right-click **Results** to view the available **Report** options. Then select a report to open the **Report** window. Refer to Creating Reports.

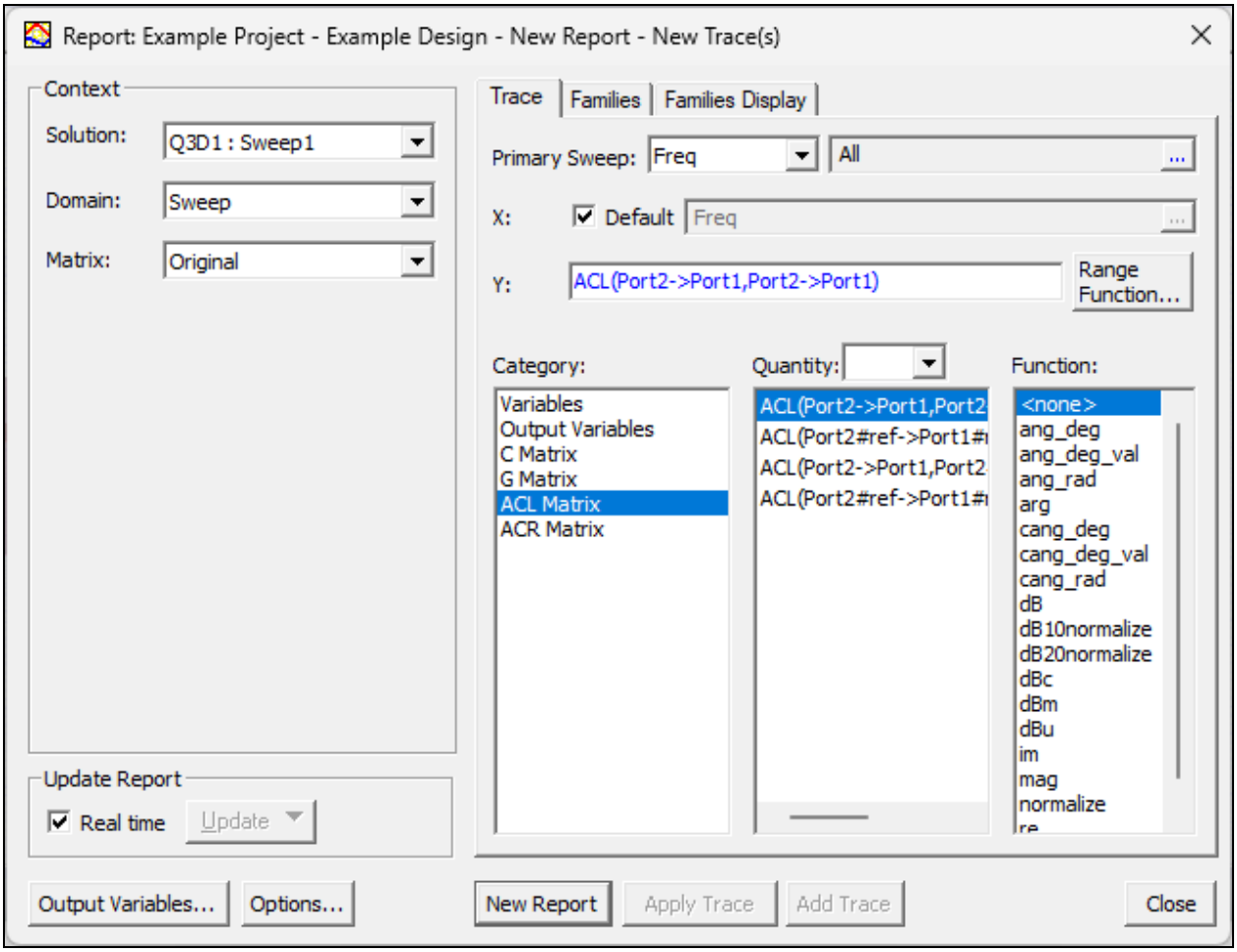
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



28. There are several report options unique to Q3D solution setups. Refer to the following:

Sweep Solution

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



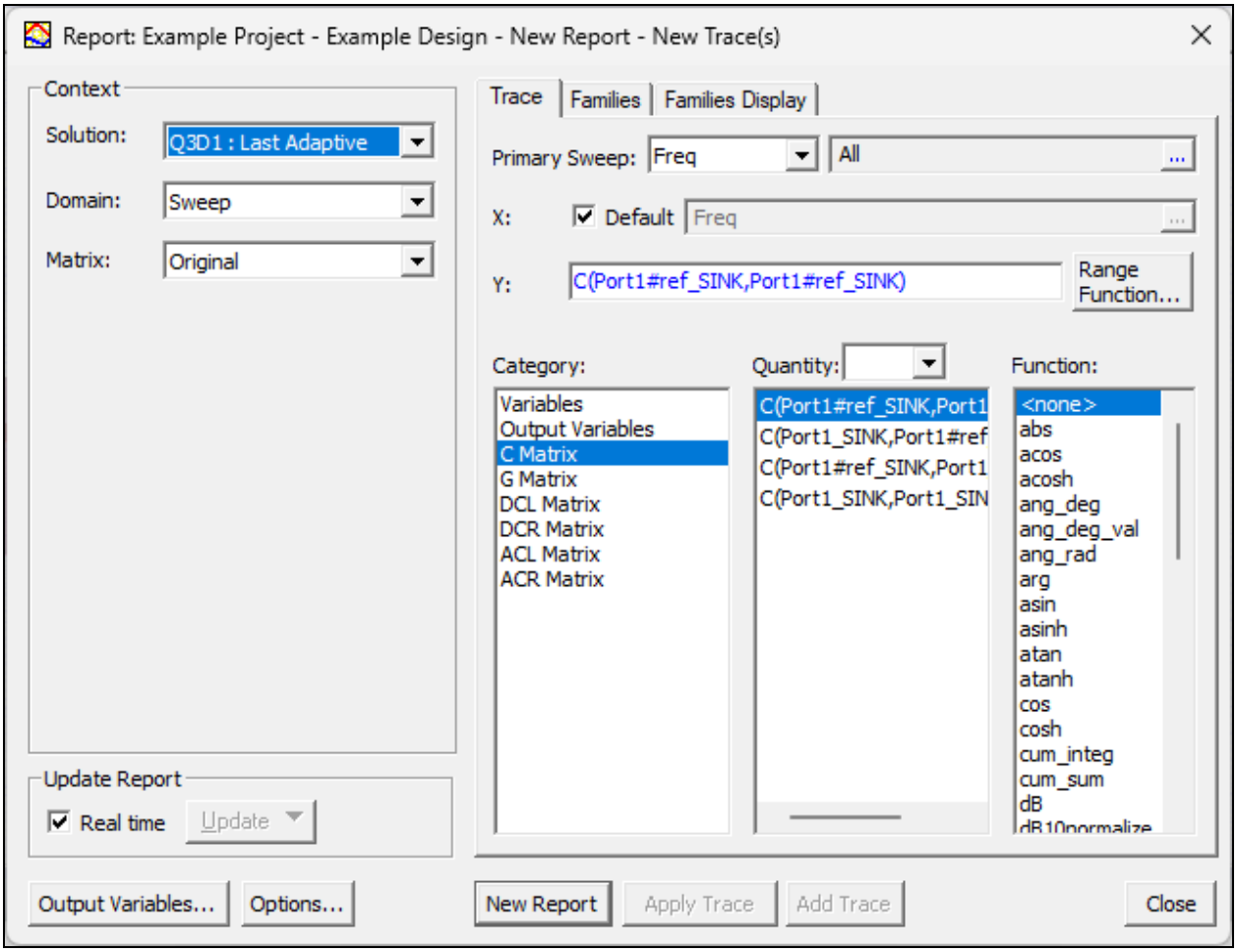
a. From the **Solution** drop-down menu, select **Sweep**.

b. From the **Matrix** drop-down menu, select one of the following:

- **Original** – populates the **Category** and **Quantity** list with **C**, **G ACL** and **ACR** matrices pertinent to the Q3D simulation.
- **Reduced** – populates the **Category** list with parameters (e.g., **S Parameter**).

Last Adaptive Solution

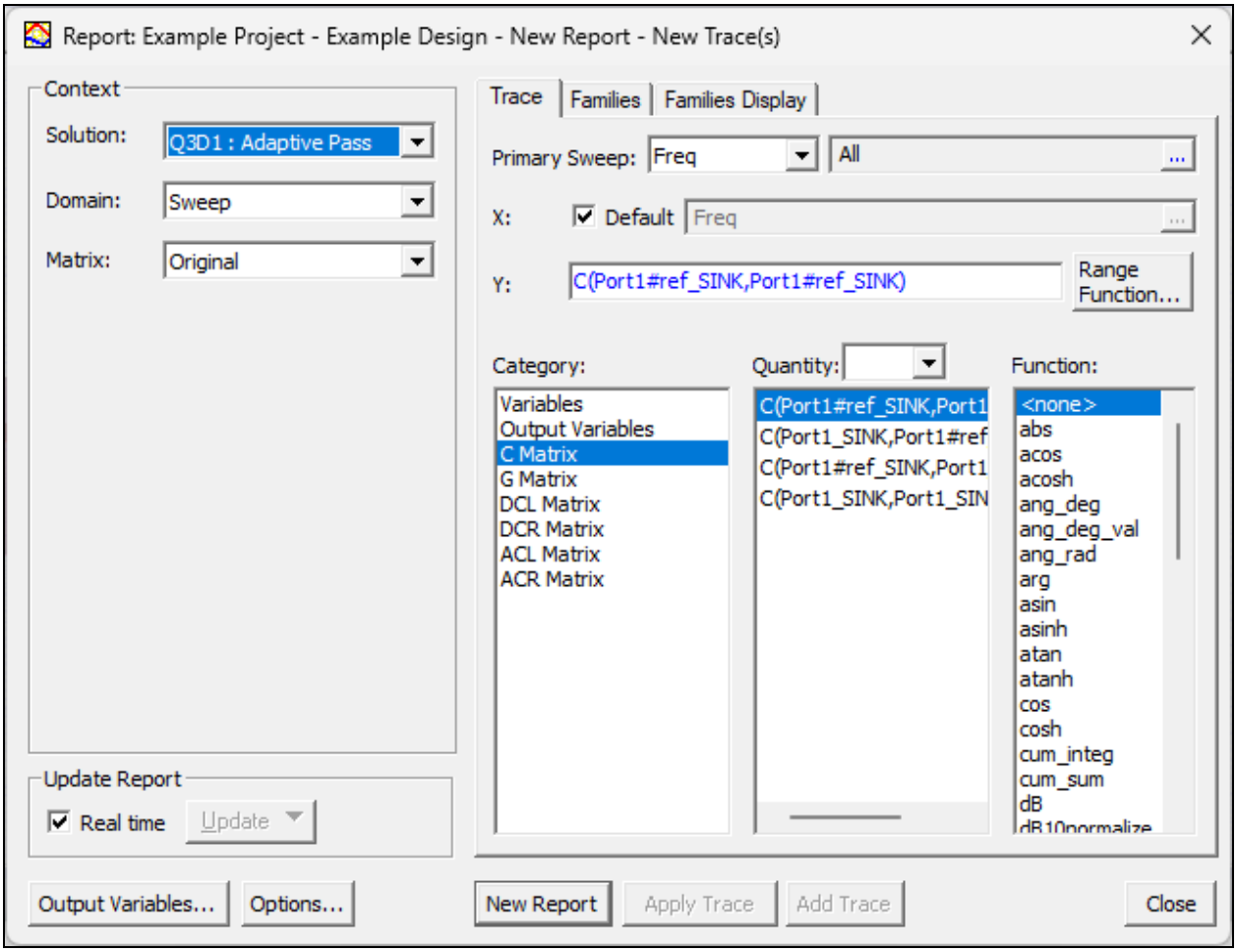
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Solution** drop-down menu, select **Last Adaptive** to populate the **Category** and **Quantity** list with matrices pertinent to the Q3D simulation (i.e., **C**, **G**, **DCL**, **DCR**, **ACL**, and **ACR**).

Adaptive Pass Solution

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

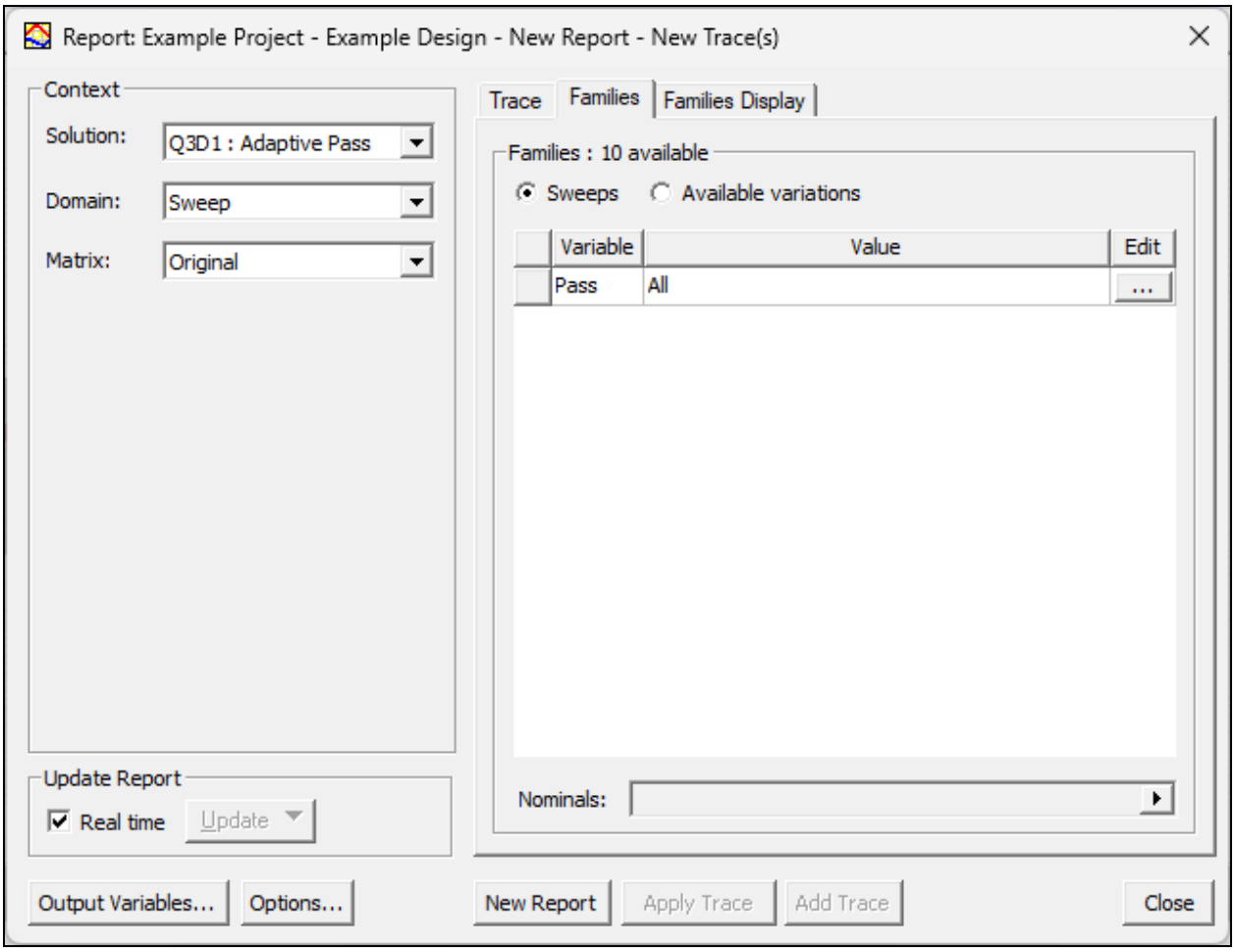


a. From the **Solution** drop-down menu, select **Adaptive Pass** to populate the **Category** and **Quantity** list with matrices pertinent to the Q3D simulation (i.e., **C**, **G**, **DCL**, **DCR**, **ACL**, and **ACR**).

b. Navigate to the **Families** tab and select either all passes or an individual pass (if

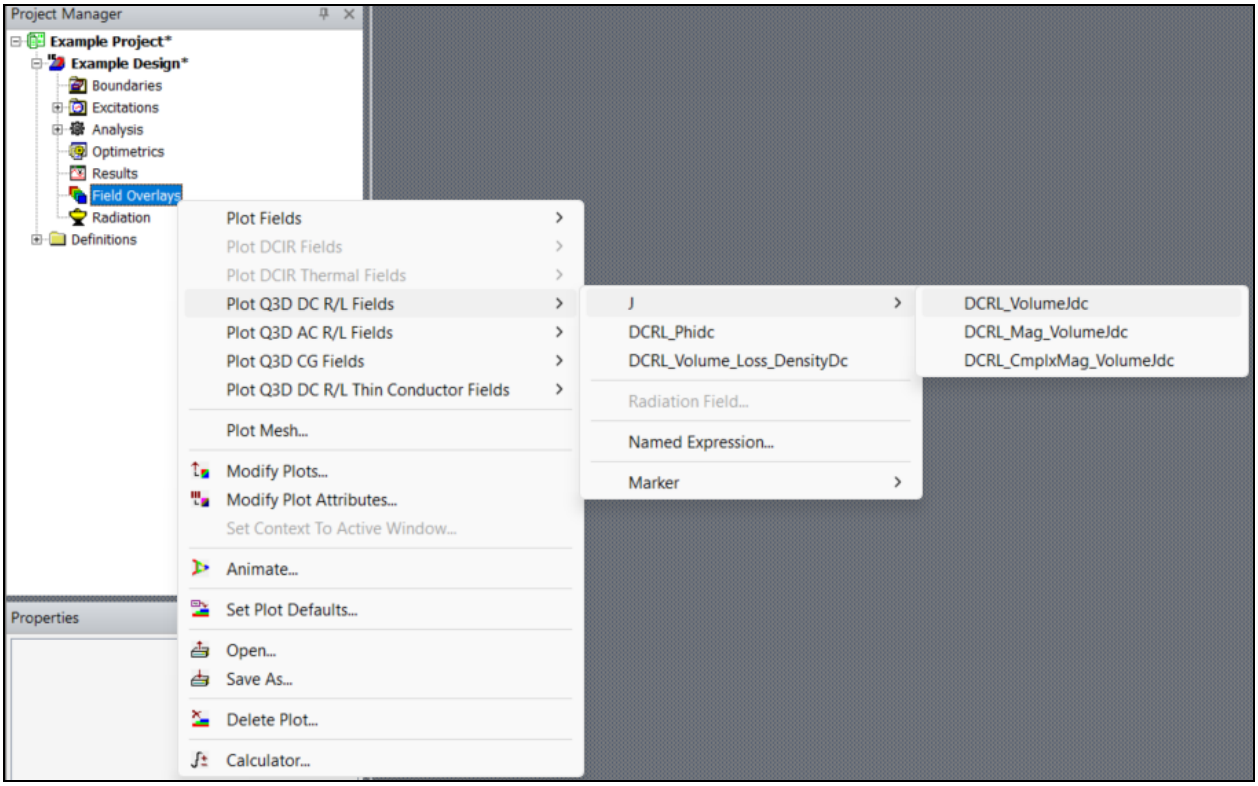
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiple sweeps are available) to view.



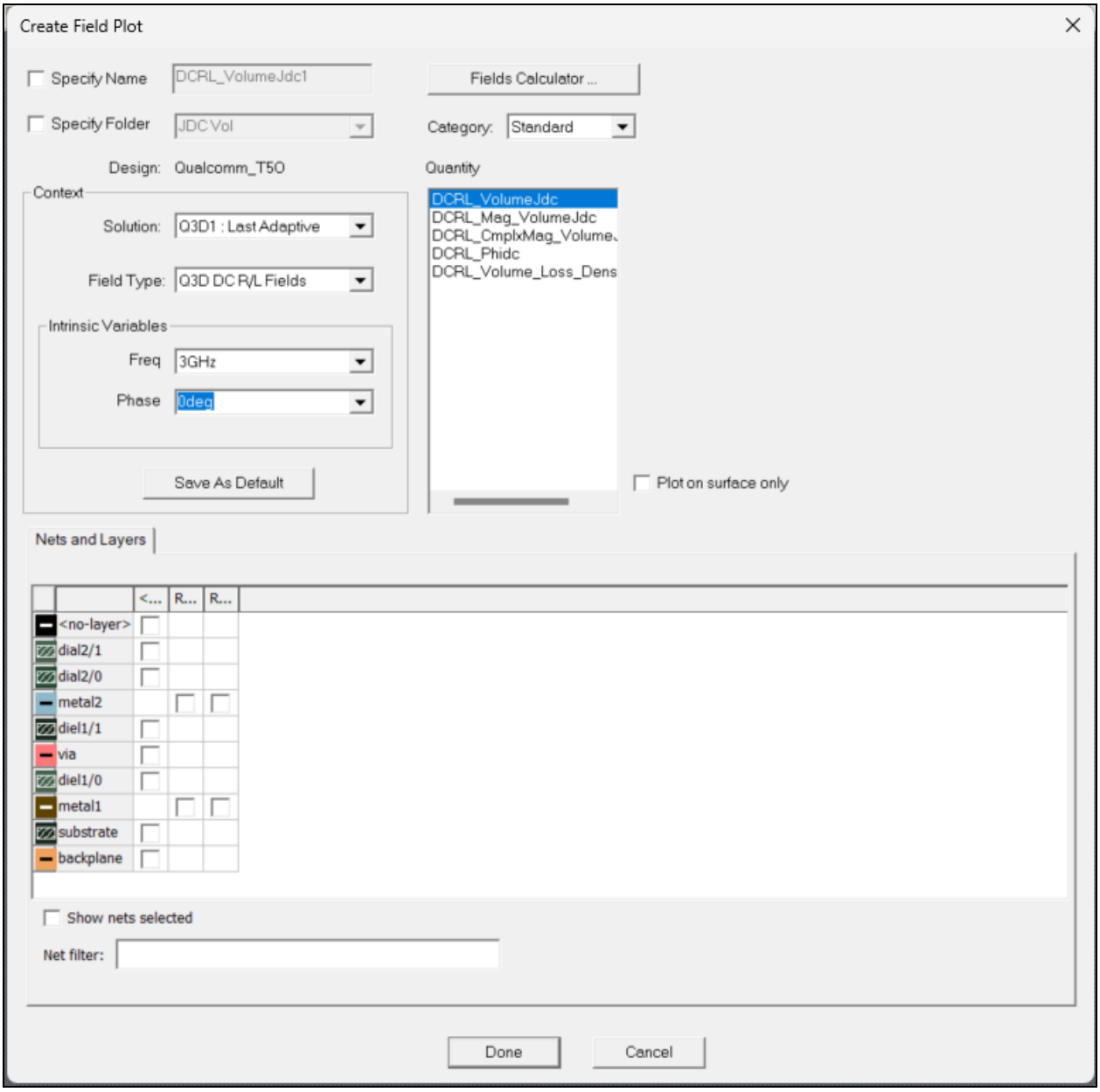
29. Navigate to the **Project Manager** window and right-click **Field Overlays** to view the available plot types. Several plot types are unique to Q3D solution setups (i.e., **NF Fields**, **DC R/L Fields**, **AC R/L Fields**, **CG Fields**, and **DC R/L Thin Conductor Fields**). Refer to Field Quantities.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



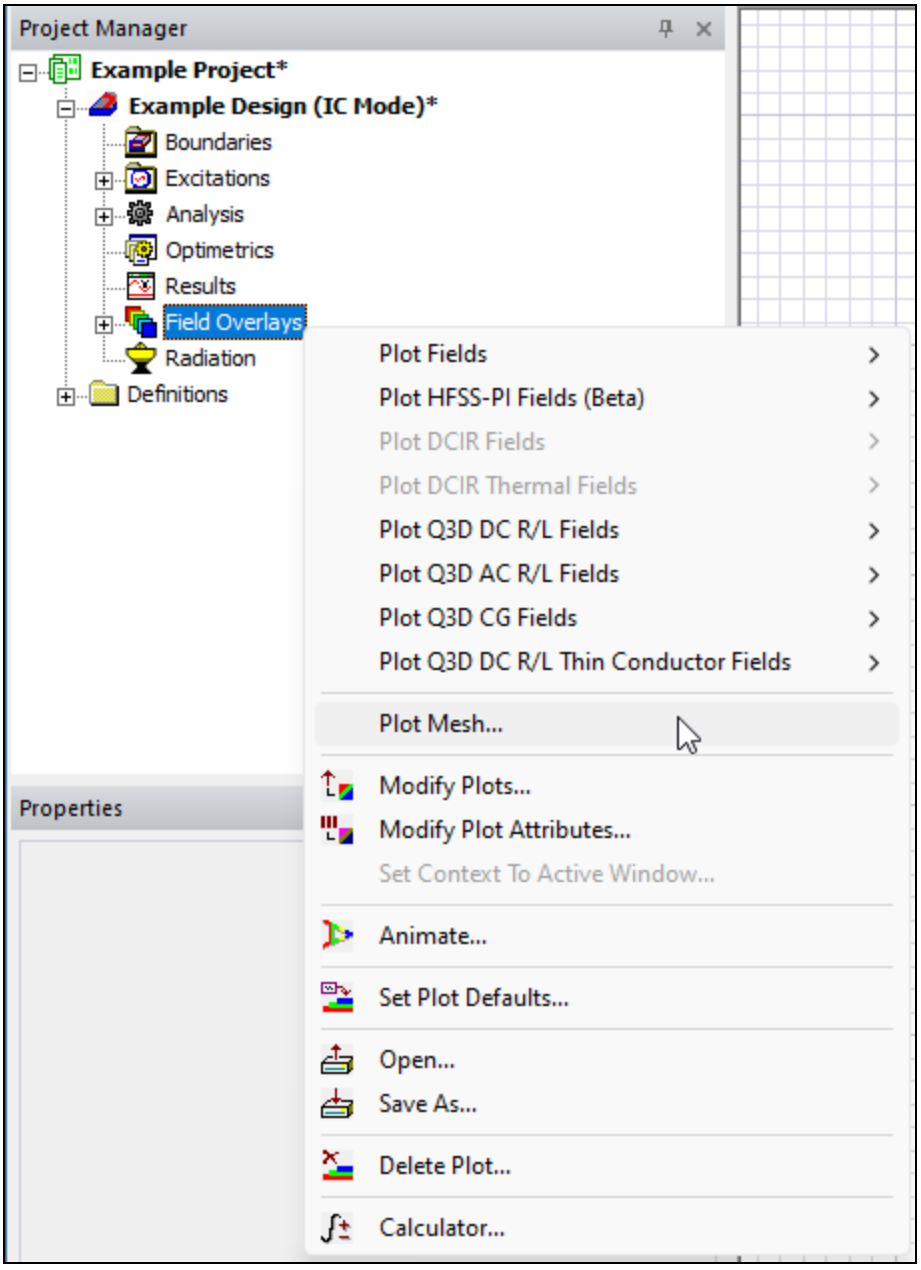
30. Select a plot type to open the **Create Field Plot** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



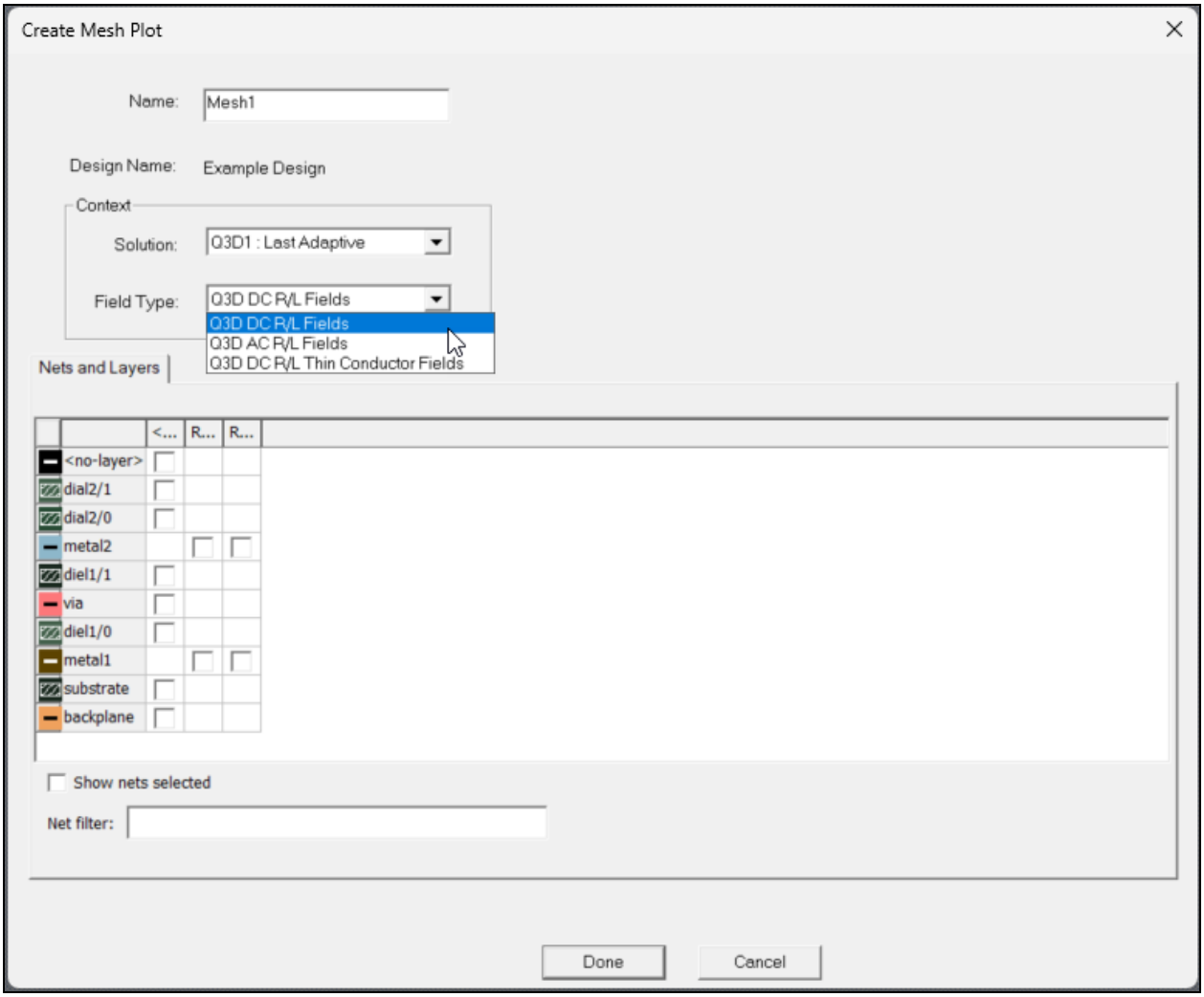
31. **Specify** a **Name**, **Folder**, and make changes to the default settings in the **Context** area and from the **Quantity** list, as appropriate. Select **Nets** and **Layers** from the table.
32. Click **Done** to create the field plot. Refer to Plotting Field Overlays.
33. Navigate to the **Project Manager** window. Then right-click **Field Overlays** and select **Plot Mesh** to open the **Create Mesh 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.



34. Select a Q3D solution from the **Solution** drop-down menu to populate the **Field Type** drop-down menu with options unique to Q3D solution setups (i.e., **DC R/L Fields**, **AC R/L Fields**, **CG Fields**, and **DC R/L Thin Conductor Fields**). Refer to Create Mesh Plot.

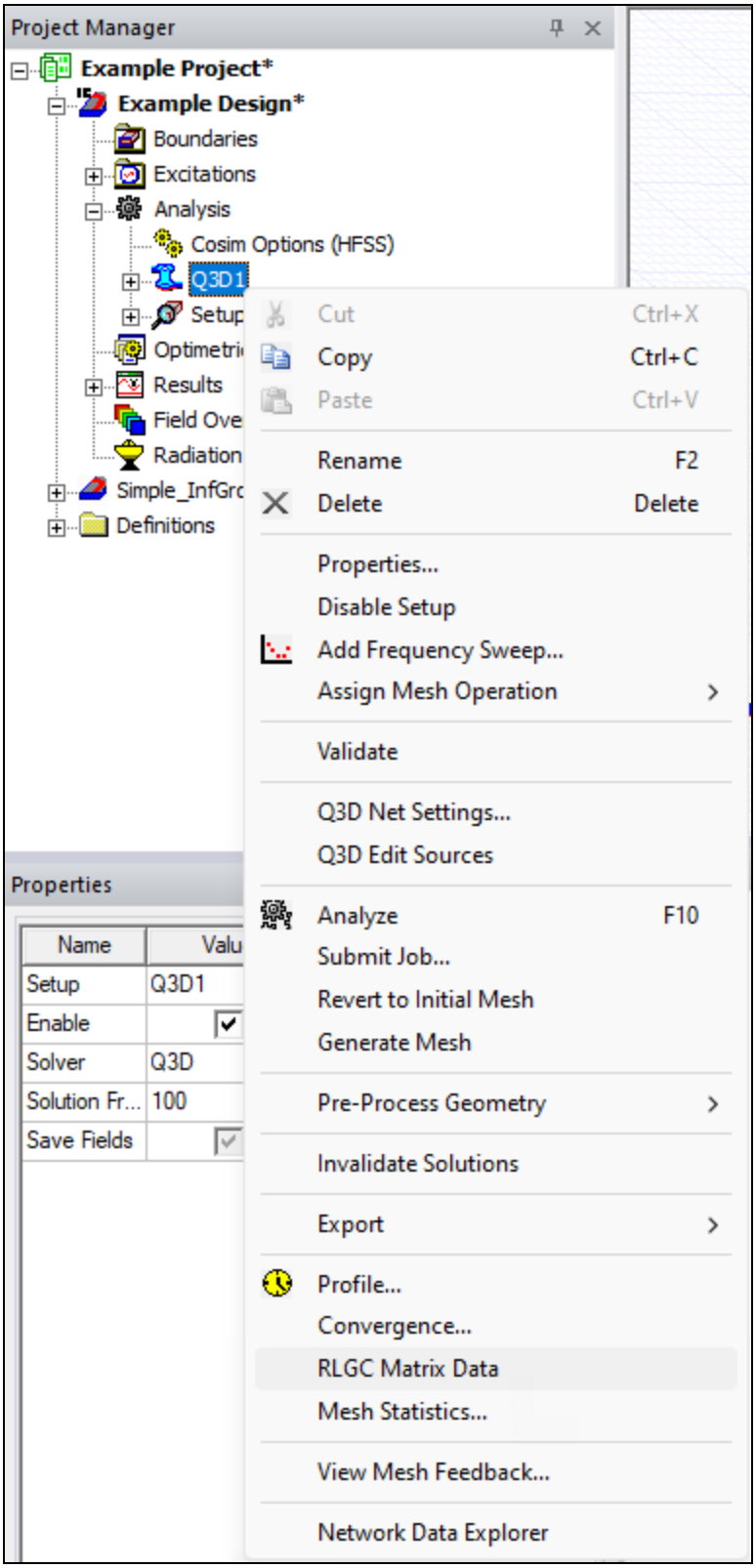
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Viewing RLGC Matrix Data

To view computed matrix data from a Q3D solution setup, navigate to the **Project Manager** window. Then right-click the Q3D solution setup (e.g., **Q3D1**) and select **RLGC Matrix Data** to open the **Solution** window. Refer to Viewing Matrix Data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



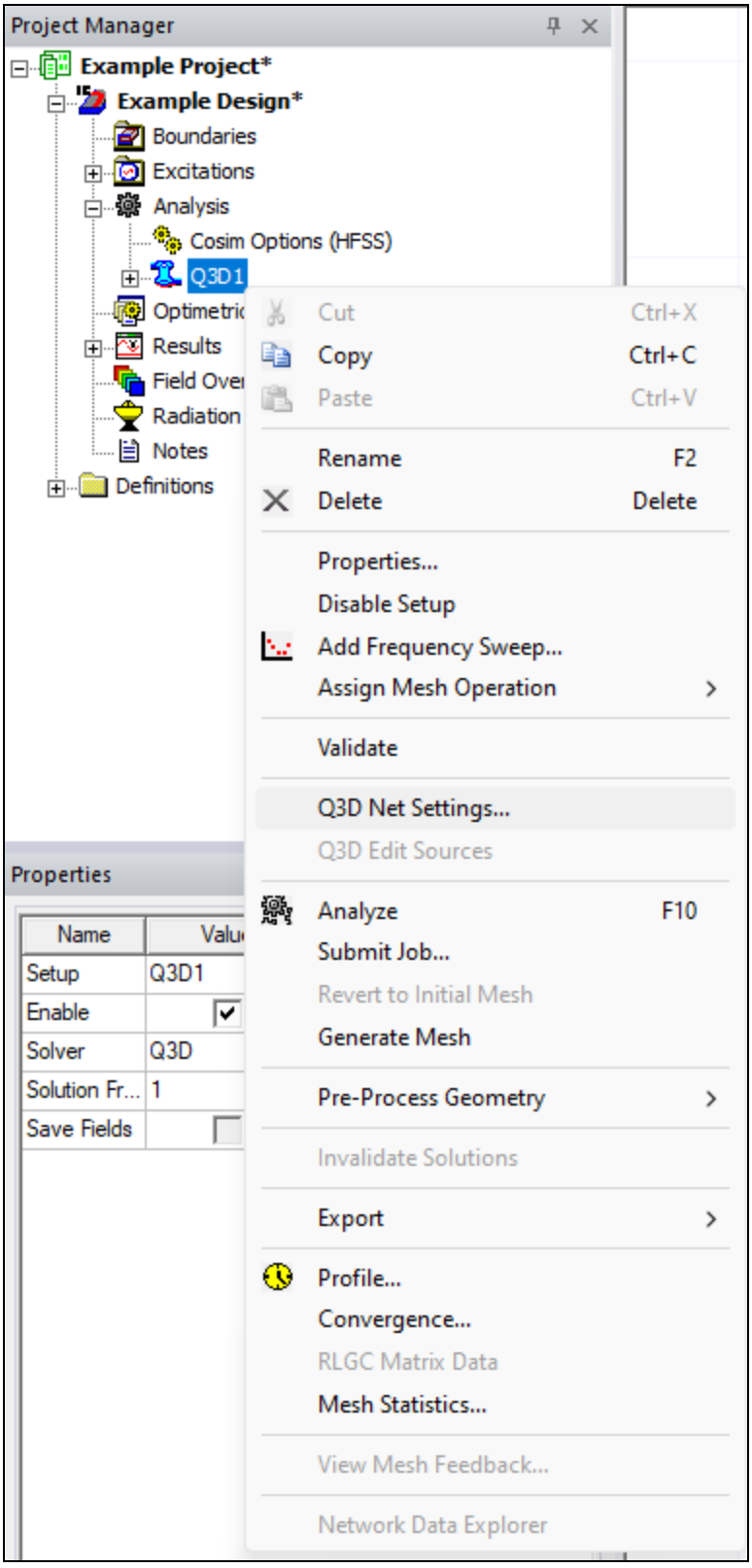
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing Net Settings

Follow these steps to view all of the nets recognized by Q3D during simulation, and modify as appropriate.

1. From the **Project Manager** window, expand **Analysis**. Then right-click the Q3D solution setup (e.g., **Q3D1**) and select **Q3D Net Settings** to open the **Q3D Net Settings** window.

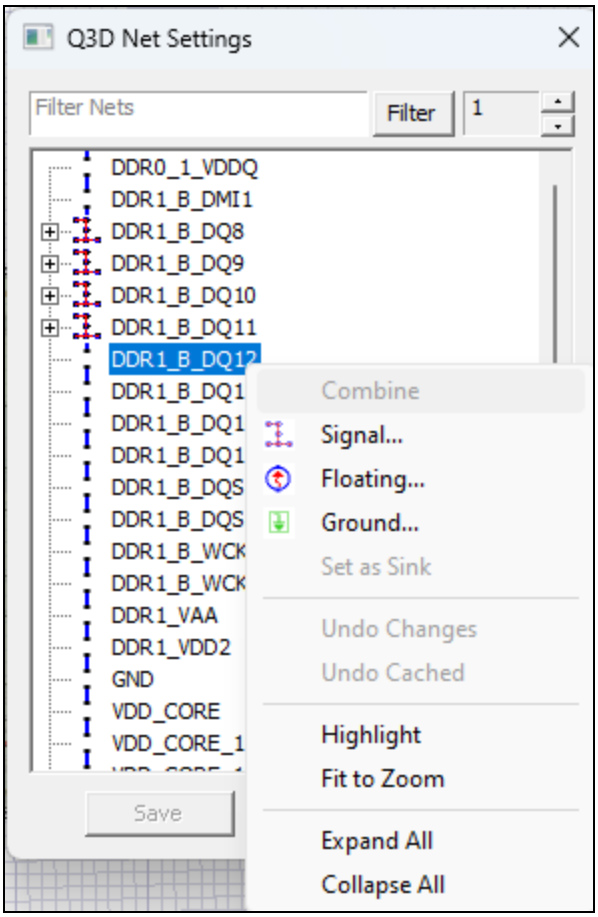
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If the mesh has not been generated by the user, AEDT will automatically generate the mesh before opening the **Q3D Net Settings** window.

2. From the **Q3D Net Settings** window, right-click the appropriate nets and/or net groups to do the following:



- **Combine** – combine (only) floating nets.
- Set floating nets to **Signal**, **Floating**, or **Ground** types.

  > **Note:**
  >
  > Regular nets can only be set to **Signal** type.

- **Set as sink** – set the selected net to **Sink** type.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Undo Changes** – restore original net attributes.
- **Undo Cached** – c1lear saved net data from the Q3D solution setup.
- **Highlight** – highlights the selected nets/sources/sinks in the **Layout Editor**.
- **Fit to Zoom** – Centers the screen on the selected nets/sources/sinks and zooms in.
- **Expand All/Collapse All** – Adjust the net node tree accordingly.

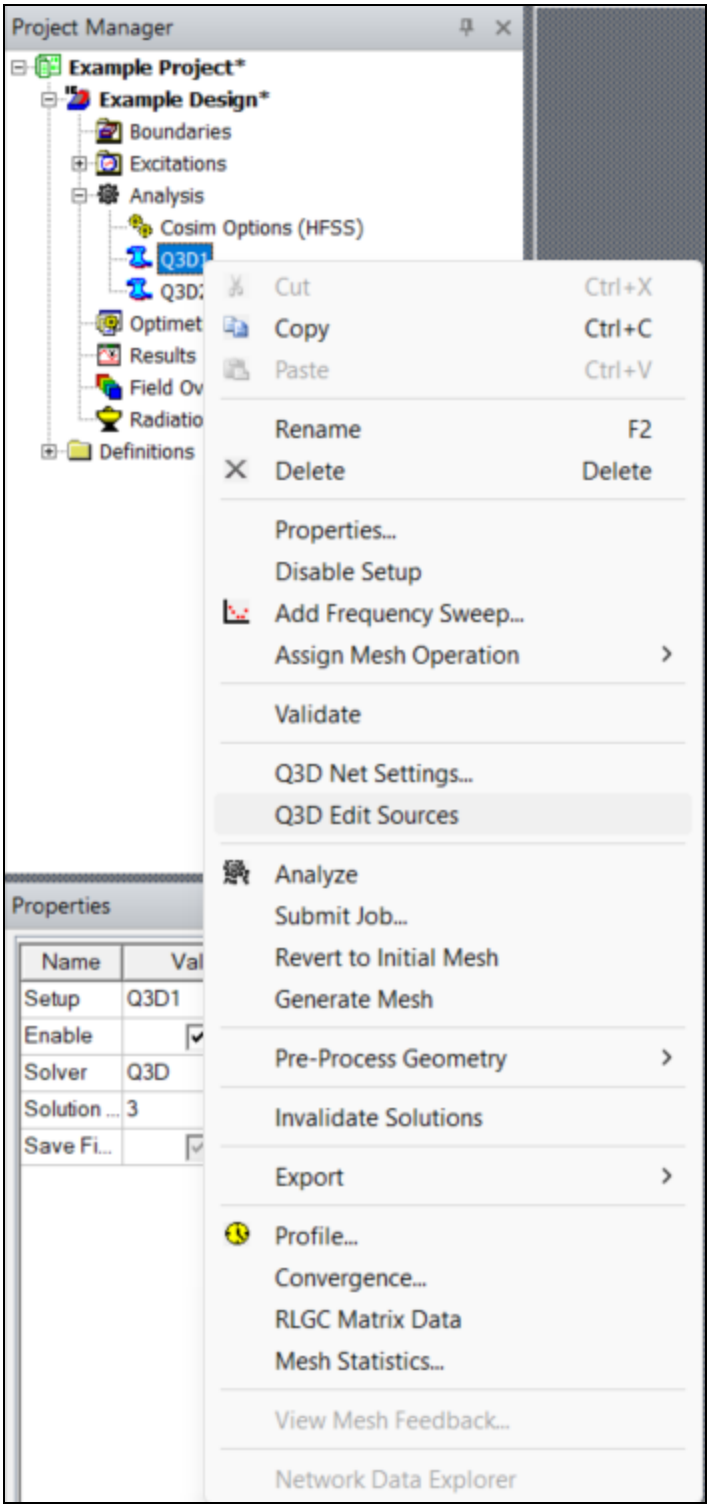
3. Click **OK** to close the **Q3D Net Settings** window.

Editing Sources

Follow these steps to edit port magnitudes and phase.

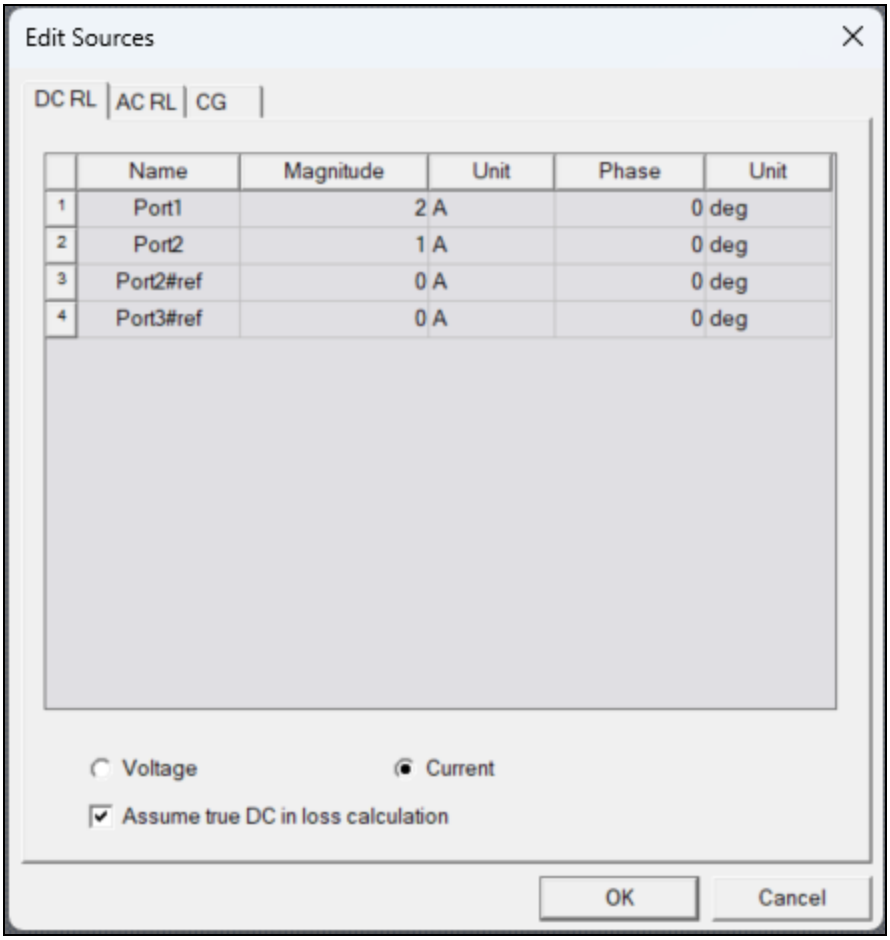
1. From the **Project Manager** window, expand **Analysis**. Then right-click the Q3D solution setup (e.g., **Q3D1**) and select **Q3D Edit Settings** to open the **Edit Sources** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Edit Sources** window, do the following from the **DC RL**, **AC RL**, and **CG** tabs:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **DC RL** – select **Voltage** or **Current** and specify those values in the table above. Manually edit the **Magnitude** and/or **Phase** values, or select a new **Unit** of measurement from the drop-down menus. Then check or uncheck **Assume true DC in loss calculation**, as appropriate.

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 **Assume true DC in loss calculation** is enabled, DC volume loss calculations assume a standard DC scenario, where currents are either fully aligned (0° degrees) or fully opposed (180° degrees). The loss is calculated as *P = IRI*. Loss calculated with this option matches legacy results.
>
> When this option is deselected, the calculation includes the possibility of low-frequency alternating components (i.e., sinusoids) with given phase shift. Both in-phase and out-of-phase aspects of the current are included, resulting in a more detailed and accurate calculation of losses, similar to how AC current losses are calculated. The loss is calculated as RMS loss:
>
> $P = \frac{1}{2} I_{re} R I_{re} + \frac{1}{2} I_{im} R I_{im}$

- **AC RL** – select **Voltage** or **Current** and specify those values in the table above. Manually edit the **Magnitude** and/or **Phase** values, or select a new **Unit** of measurement from the drop-down menus. AC RL voltage and current sources are assumed as peak quantities.
- **CG** – select **Voltage** or **Current** and specify those values in the table above. Manually edit the **Magnitude** and/or **Phase** values, or select a new **Unit** of measurement from the drop-down menus.

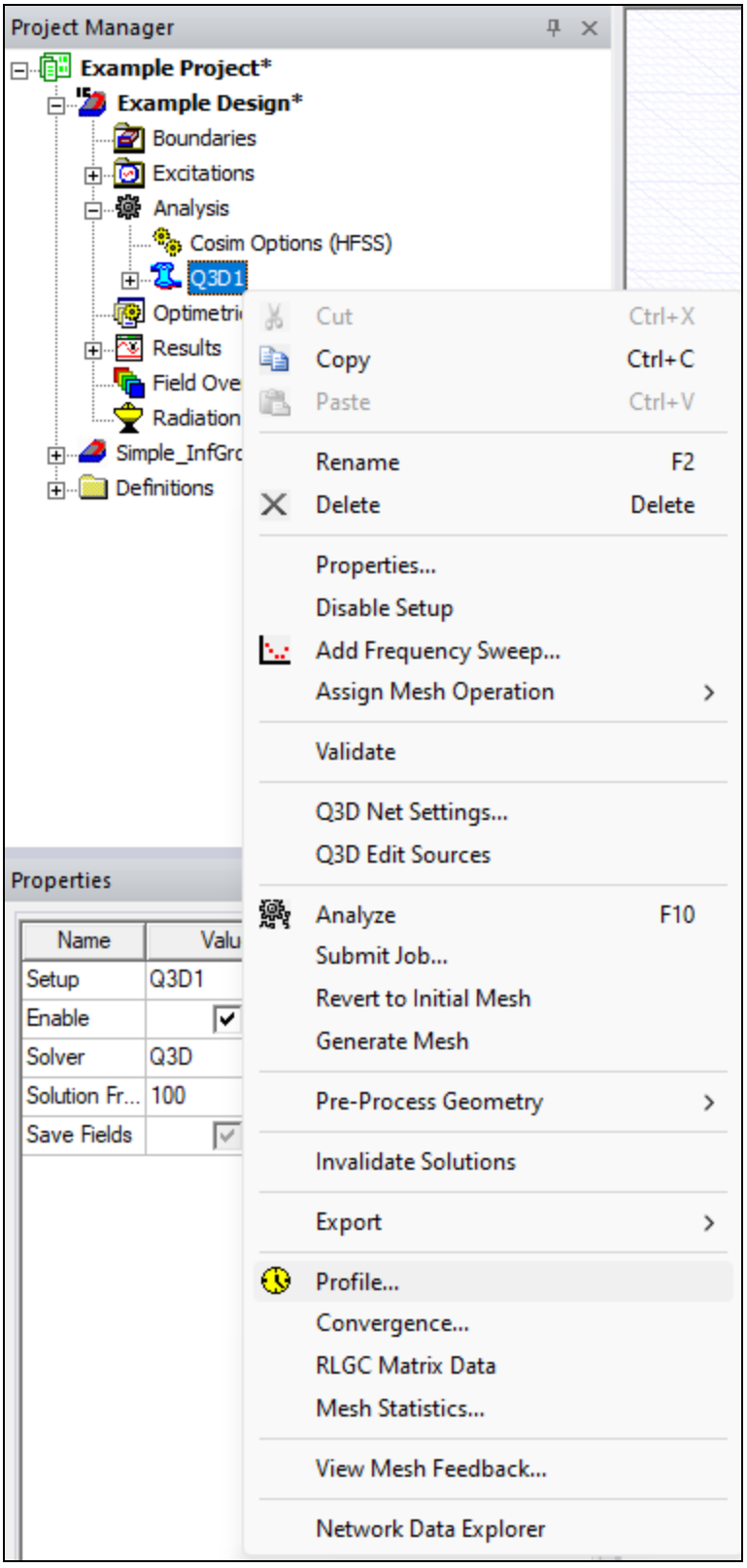
3. Click **OK** to close the **Edit Sources** window.

Exporting RLGC Matrix Data

Follow these steps to export RLGC matrix data or lumped RLGC data (i.e., **Equivalent Circuit Export**) from an analyzed solution.

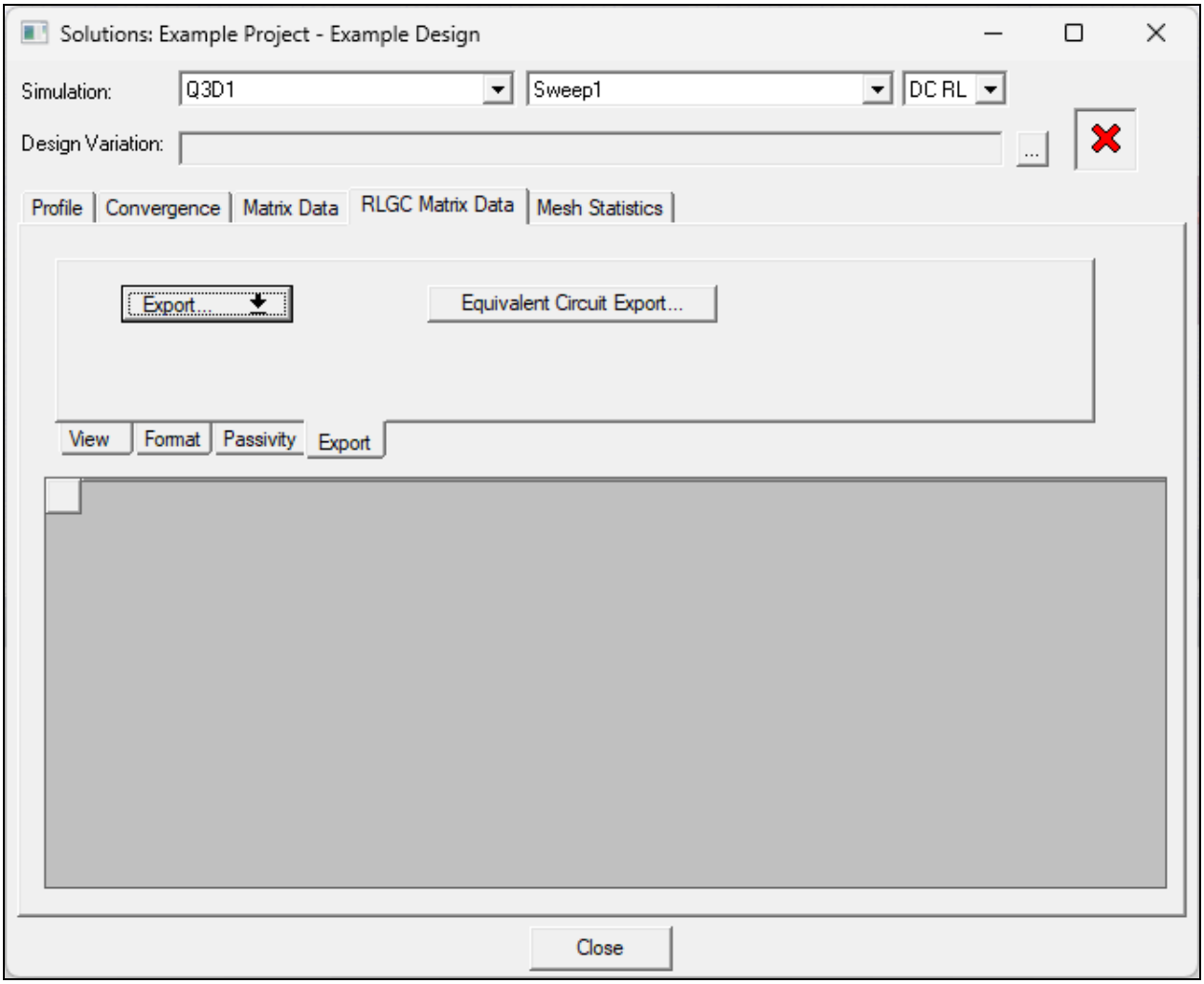
1. From the **Project Manager** window, expand **Analysis**. Then right-click the Q3D solution setup (e.g., **Q3D1**) and select **Profile** to open the **Solutions** window.

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.

2. From the **Solutions** window, navigate to the **RLGC Matrix Data** tab.
3. Select the **Export** subtab.

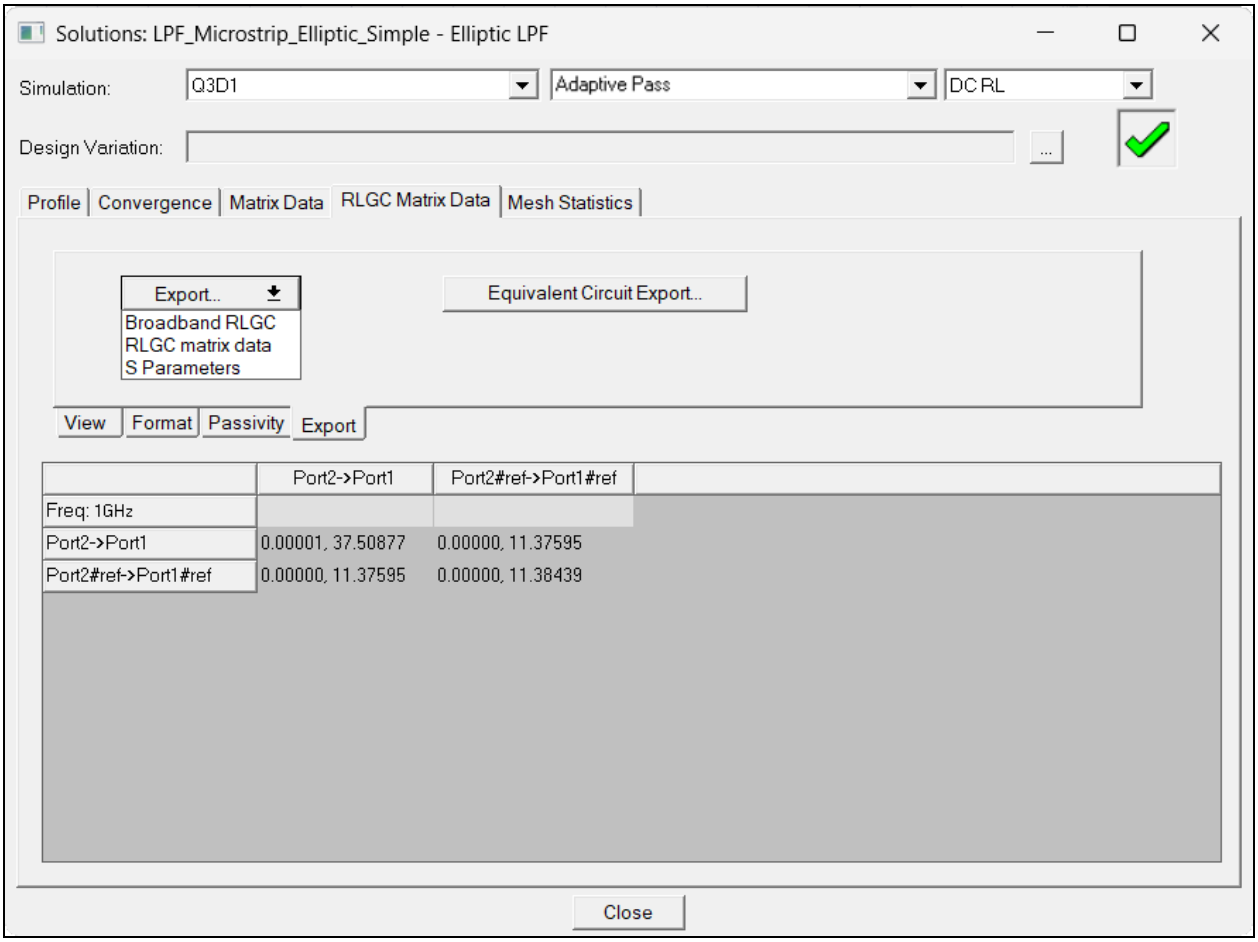


4. To export RLGC data in one or more of several file formats, or to export lumped RLGC data as an equivalent circuit file, complete steps in the following subsections.

Exporting RLGC Data in Several Formats

To export RLGC data, select **Broadband RLGC**, **RLGC matrix data**, or **S Parameters** from the **Export** drop-down menu.

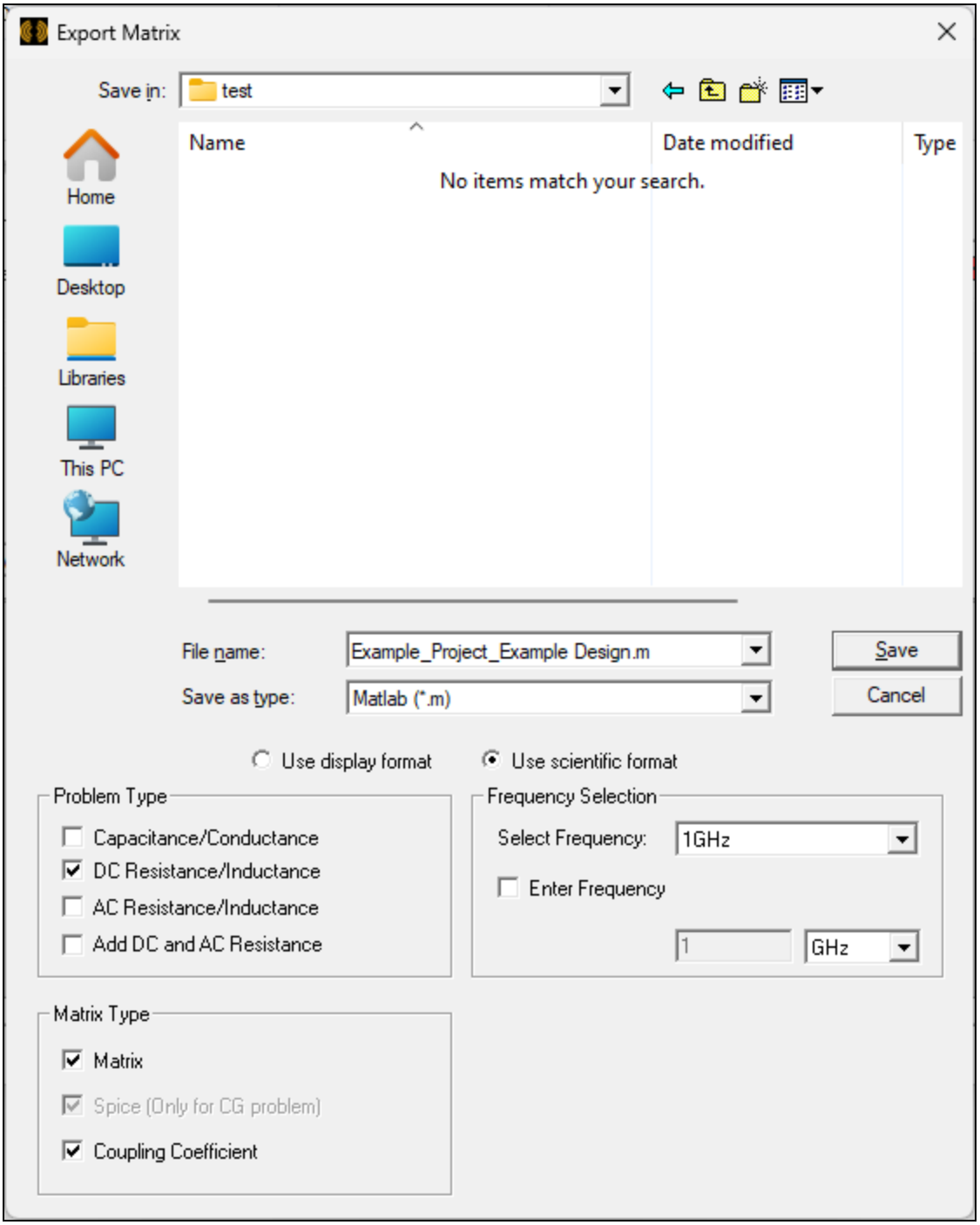
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Broadband RLGC** — generates a netlist that incorporates a causal RLGC model, ensuring physically realistic frequency dependence, and replaces the standard RLGC subcircuit with a state-space model.
- **RLGC matrix data** — produces a frequency-dependent RLGC data matrix that contains resistance (R), inductance (L), conductance (G), and capacitance (C) values used to model transmission lines.
- **S-Parameters** — outputs an S-parameter file that characterizes the circuit's response to different frequencies.

Depending on the selection, a **Save As** or **Export Matrix** window appears. To save a Broadband RLGC model, navigate to an appropriate directory, enter a **File name**, select a file type from the **Save as type** drop-down menu, and click **Save**.

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 a matrix of RLGC data, do the following:

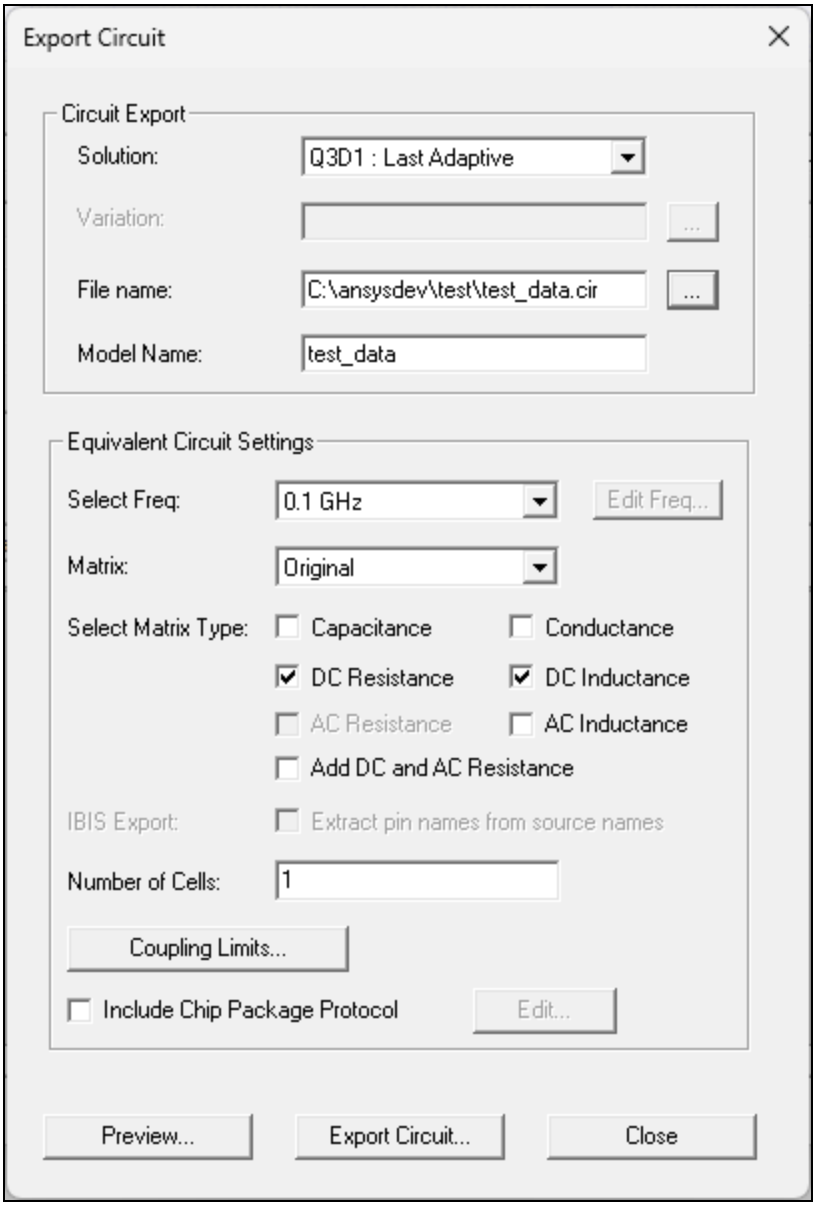
1. Enter a **File name** or accept the default name.
2. Select a file type from the **Save as type** drop-down menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the **Problem Type**, **Frequency Selection**, and **Matrix Type** areas, choose the RLGC data to export.
4. Click **Save** to export the data and close the window.

Exporting Lumped RLGC Data in an Equivalent Circuit File Format

1. To export lumped RLGC data, click **Equivalent Circuit Export** to open the **Export Circuit** 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. From the **Circuit Export** area, specify the following:

    - **Solution** – use the drop-down menu to select the solution from which to export data (e.g., **Q3D1 : Last Adaptive**).
    - **Variation** – click **...** to select a variation, if any are available.
    - **File Name** – enter the file path and name of an appropriate circuit file (e.g., **C:\ansysdev\test\test_data.cir**) or click **...** to open an explorer window, then navigate to an appropriate directory, enter a **File name**, and click **Open**.

      Important:

      Circuit data can be exported in any of the following formats:

      - **Simplorer Netlist Files (*.sml)**
      - **HSPICE Circuit (*.sp)**
      - **IBIS Package Model (*.pkg)**
      - **Maxwell Spice Circuit (*.spc)**
      - **PSpice Circuit (*.lib)**
      - **Spectre Circuit (*.ckt)**
      - **Ansoft Designer Netlist (*.cir)**
      - **Berkeley Spice Circuit (*.bsp)**
      - **Cadence DML (*.dml)**
      - **IBIS ICM (*.icm)**

      To export circuit data as an IBIS Package Model, use all lowercase names to facilitate portability between operating systems. The length should not be more than 20 characters.

      To export circuit data in IBIS ICM format, use all lowercase names to facilitate portability between operating systems. The extension should be no more than three characters in length. There is no length restriction on the base file name; however, only lowercase letters, digits 0 through 9, underscore and hyphen are valid characters.

    - **Model Name** – enter a name for the model within the circuit file (e.g., **test_data**). Upper and lowercase letters, digits 0 through 9, and underscore are valid characters. Spaces are not accepted. Invalid characters will be replaced with underscores.

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 **Equivalent Circuit Settings** area, specify the following:

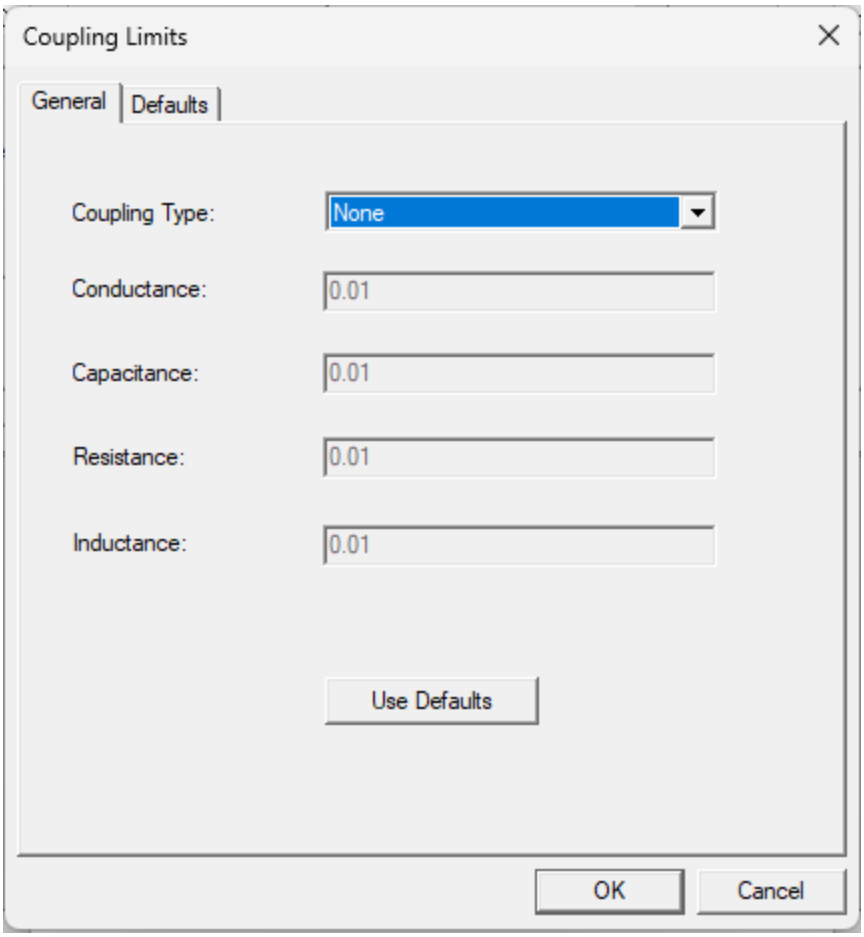
   - **Select Freq** – select a frequency from the drop-down menu (e.g., **0.1 GHz**). For interpolating sweeps, click **Edit Freq** to open the **Edit Sweep** window, then enter **Start** and **End** values.
   - **Matrix** – from the drop-down menu, select the **Original** solved matrix or a reduced matrices, if any are available.
   - **Select Matrix Type** – check the check boxes next to one or more matrix types for export (i.e., **Capacitance**, **Conductance**, **DC Resistance**, **DC Inductance**, **AC Resistance**, **AC Inductance**, and/or **Add DC and AC Resistance**).

     > **Note:**
     >
     > Checking the **DC Resistance** or **DC Inductance** boxes deactivates the boxes beside **AC Resistance** or **AC Inductance**, and vice versa. Check the **Add DC and AC Resistance** box to create a circuit with a total resistance computed from AC and DC values.

   - **IBIS Export** – by default, names are created in the form **_NAME_src**. Checking the **Extract pin names from source names** box will alter naming behavior so they are created as simply **NAME**.
   - **Number of Cells** – enter the number of cells (i.e., sections) to use while exporting the circuit. Finer discretization (i.e., a larger number of sections) gives a closer approximation to the underlying **Partial Differential Equations**. However, this also leads to increased simulation time. See the technical notes for Equivalent Circuits.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

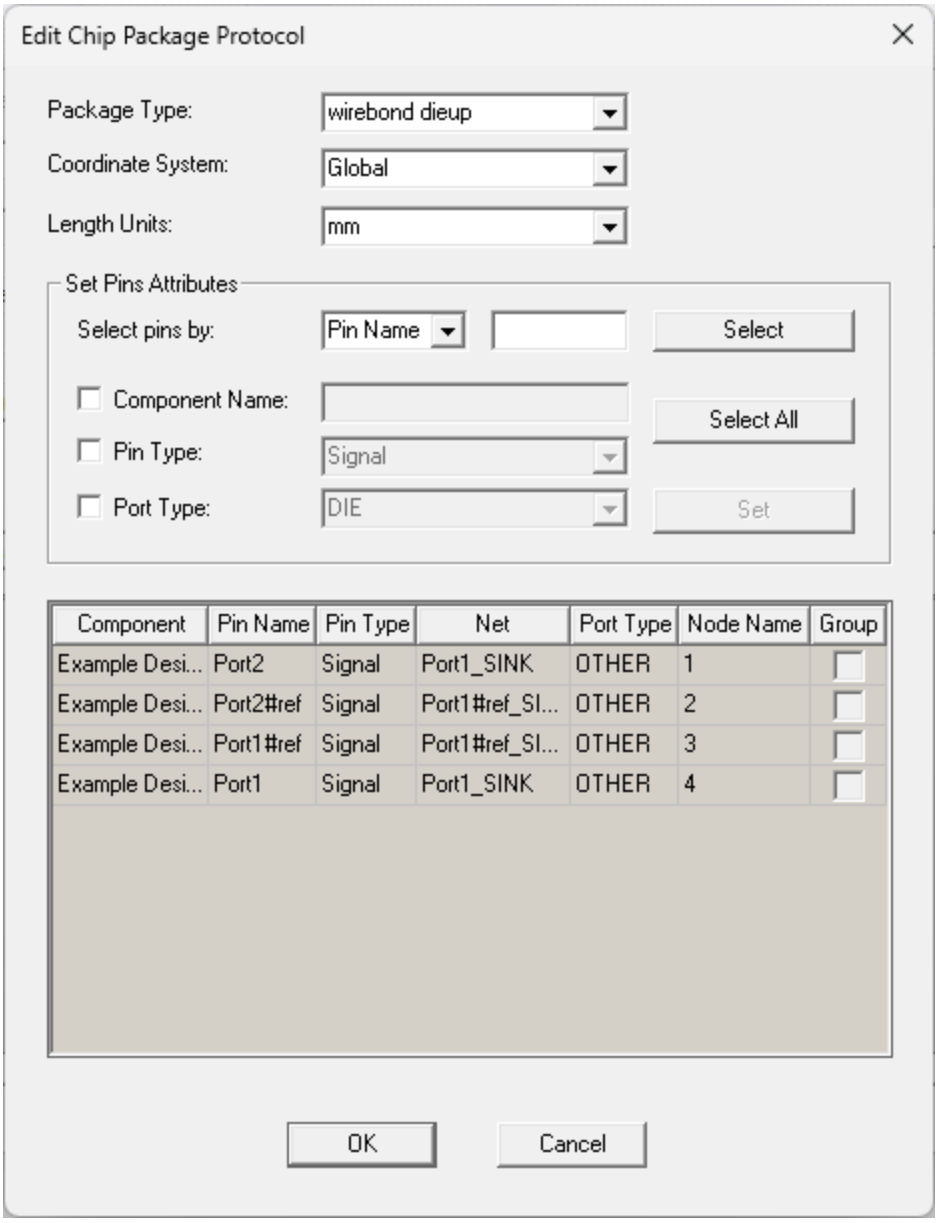
- **Coupling Limits** – click to open the **Coupling Limits** window.



  Select a **Coupling Type**, if appropriate (i.e., **By Value** or **By Fraction of Self Term**) and enter limit values in the **Conductance**, **Capacitance**, **Resistance**, and **Inductance** fields. RLCG values smaller than those specified will be ignored during export. Click **OK** to return to the **Export Circuit** window.

- **Include Chip Package Protocol** – check the box to activate **Chip Package Protocol**, then click **Edit** to open the **Edit Chip Package Protocol** window and specify header information for SPICE formats.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Chip Package Protocol (CPP) is a set of statements specifying the location, net information, and SPICE node name for each package pin (i.e., the landing pad of flip-chip bumps or the chip landing pad of bonding-wires). The CPP is embedded within a SPICE-compatible file header as comments and is primarily used to help automate the co-simulation of IC (i.e., die) and package or package/board (e.g., utilizing CPP to import the package or package and PCB model into the full chip dynamic voltage drop analysis, and the global IO-SSO analysis. Package/PCB simulation tool can utilize the CPP to import the chip model in the system level

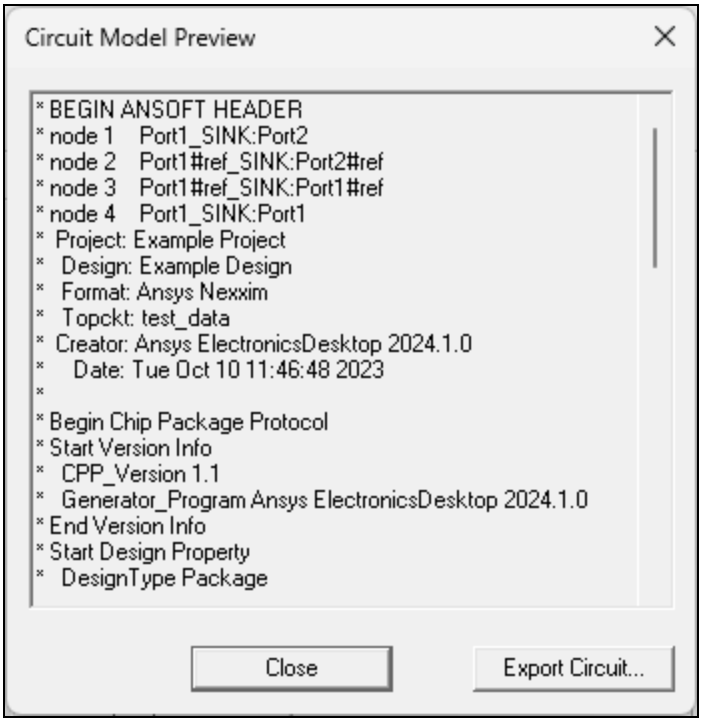
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

analysis). To utilize CPP, specify the following from the **Edit Chip Package Protocol**window:

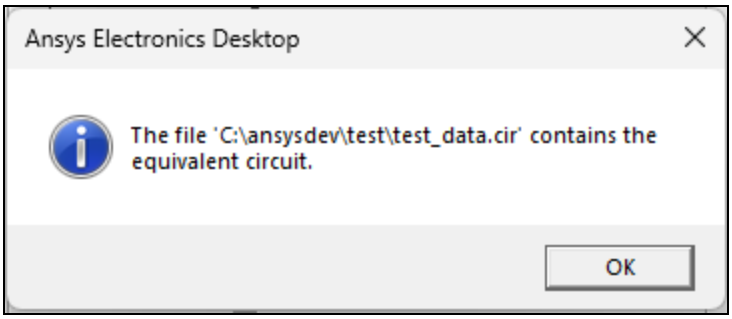
- **Package Type** – select a **Package Type** from the drop-down menu (i.e., **wirebond dieup**, **wirebond diedown**, **flipchip dieup**, and **flipchip diedown**.
- **Coordinate System** – from the drop-down menu, select **Global** or a custom coordinate system, if any are available.
- **Length Units** – Select a unit of measurement from the drop-down menu (i.e., **um**, **mm**, or **mil**).
- From the **Set Pins Attributes** area, specify the following to select pins:
  - **Select pins by** – select a sort attribute from the drop-down menu to sort pins by **Component Name**, **Pin Name** (i.e., number), **Pin Type** (i.e., signal or power ground), **Net Name**, or **Port Type** (i.e., DIE, PCB, VRM, or Other). Then enter a regular expression in the field and click **Select** to select the applicable pins.
  - **Select All** – click to select all pins in the design.
  - Check the **Component Name**, **Pin Type**, and **Port Type** boxes to modify the information for selected pin(s). Enter the **Component Name** in the field and/or select a **Pin Type**/**Port Type** from the drop-down menus.
  - Click **Set** to apply changes.

4. Click **OK** to close the **Edit Chip Package Protocol** window.
5. Click **Preview** to open the **Circuit Model Preview** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



14. Click **Close**.
15. Click **Export Circuit** to complete the export. Once the export is finished, a message appears.



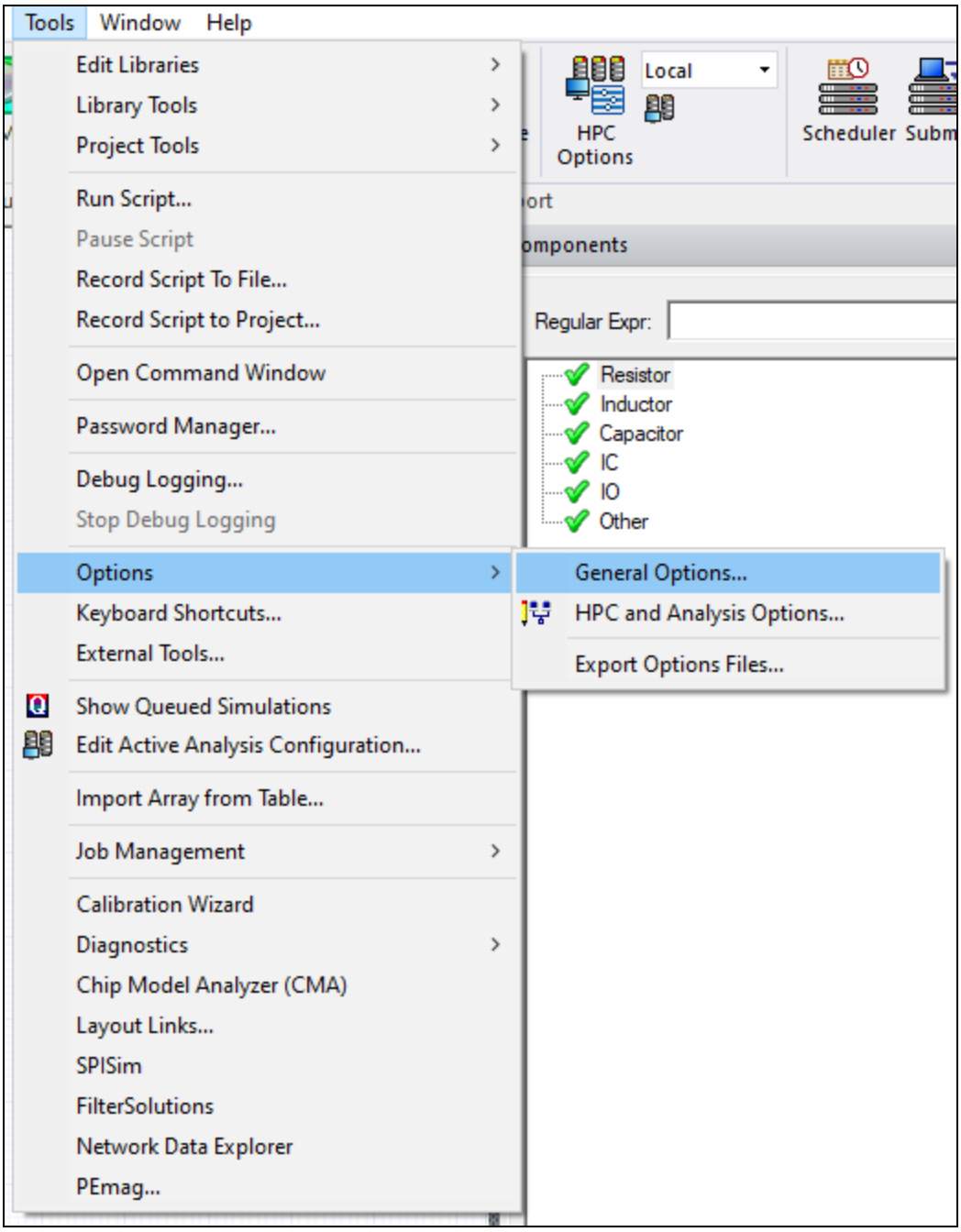
16. Click **OK** to close the message.
17. Click **Close** to exit the **Export Circuit** window.

Activating the Q3D Solution Setup Beta Feature

Follow these steps to enable the Q3D solution setup beta feature in General mode.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

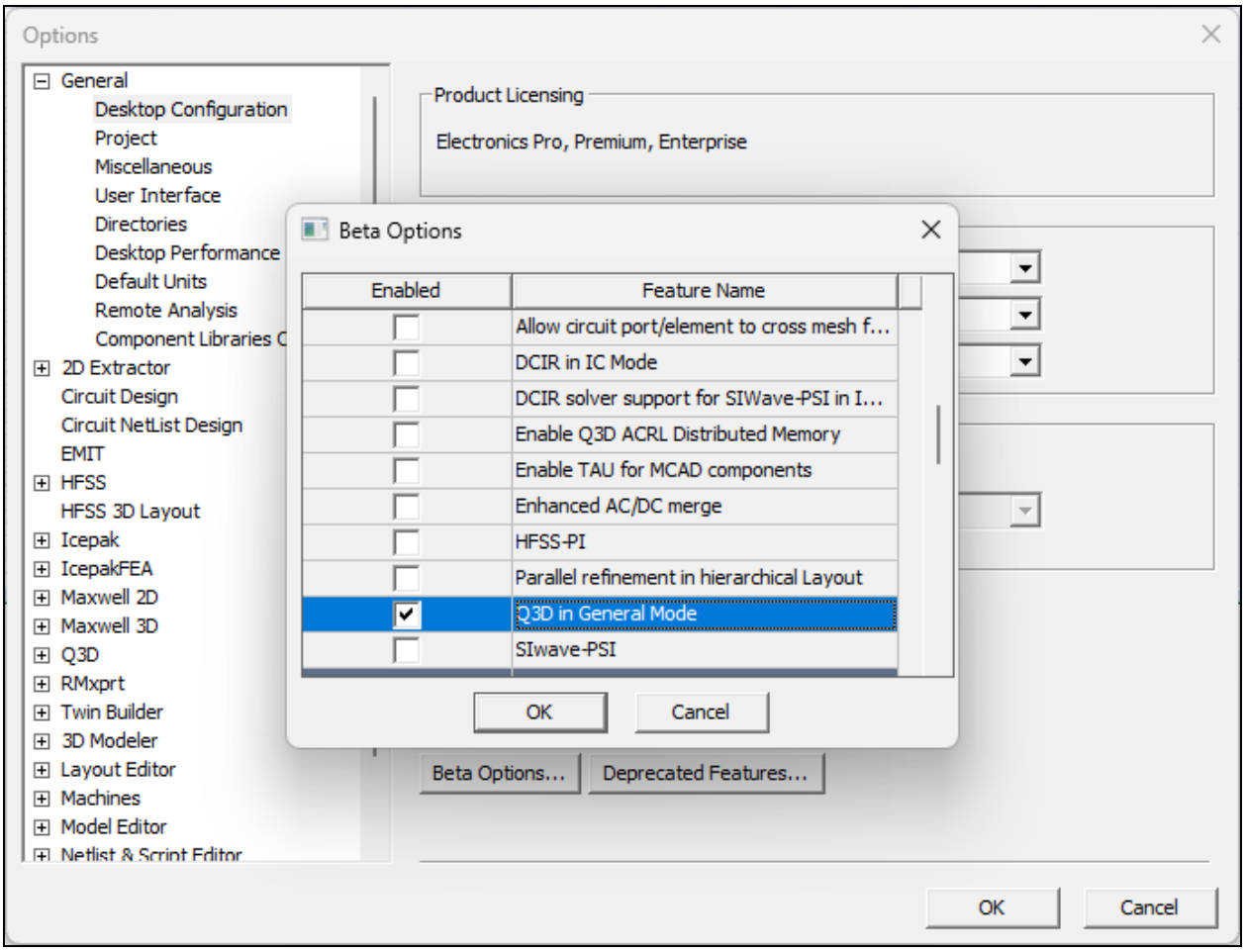
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window



2. Expand **General** > **Desktop Configuration** and click **Beta Options** to open the **Beta Options** window

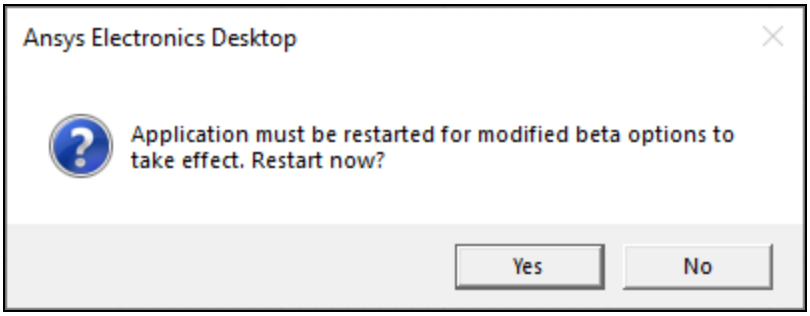
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scroll down to the **HFSS 3D Layout** section and check the box adjacent to **Q3D in General Mode**.



4. Click **OK** to close the **Beta Options** window.
5. Click **OK** to close the **Options** window and open a prompt to restart the software. Users may be asked to save their work.

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 **Yes** to restart immediately.
7. After restarting the software, the Q3D solution setup is available from the locations described in the previous section.

The procedure is complete.

Ansys SIwave™ Solutions in HFSS 3D Layout

Ansys SIwave™ is a printed circuit board (PCB) and package electromagnetics simulation software for power-ground plane and signal integrity analysis. It employs a powerful full-wave analysis engine to generate both frequency-domain and time-domain results. Using the highly optimized internal field solver, SIwave simulates complete board/package designs with full-wave accuracy. Simulation results can be reported graphically using advanced 2D and 3D plotting methods. For more information about the standalone SIwave product, consult SIwave help.

> **Note:**
>
> Not all solution types are available through the IC Mode environment.

HFSS 3D Layout contains the following SIwave solution types:

- SIwave SYZ Solver – a high-speed hybrid solver for medium and large-scale analysis of PCBs and packages, the SIwave SYZ Solver calculates frequency-dependent S-parameters for signal integrity and EMI investigations, and board-level power supply impedance modeling.
- Near Field Solver – a solver for analyzing the near-field region of PCBs and packages, which calculates electromagnetic fields and currents to assess near-field EMI, crosstalk, and impedance effects, ensuring optimal signal integrity and electromagnetic compatibility (EMC).
- Far Field Solver – a solver for analyzing the far-field region of PCBs and packages, which calculates electromagnetic radiation patterns and scattering effects to assess far-field EMI, antenna performance, and regulatory compliance, ensuring optimal signal integrity and electromagnetic compatibility (EMC).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- SIwave DC IR Solver – a solver for voltage drop across the power plane and provides important information pertaining to power delivery.
- SIwave-PSI Solver – a solver that enables customizable simulation of PCB, package, and RDL designs with enhanced modeling options for conductor roughness, bond wires, and dielectric fill refinement. It supports both discrete and interpolating frequency sweeps, adaptive sampling, and optional DCIR integration for comprehensive signal and power integrity analysis.

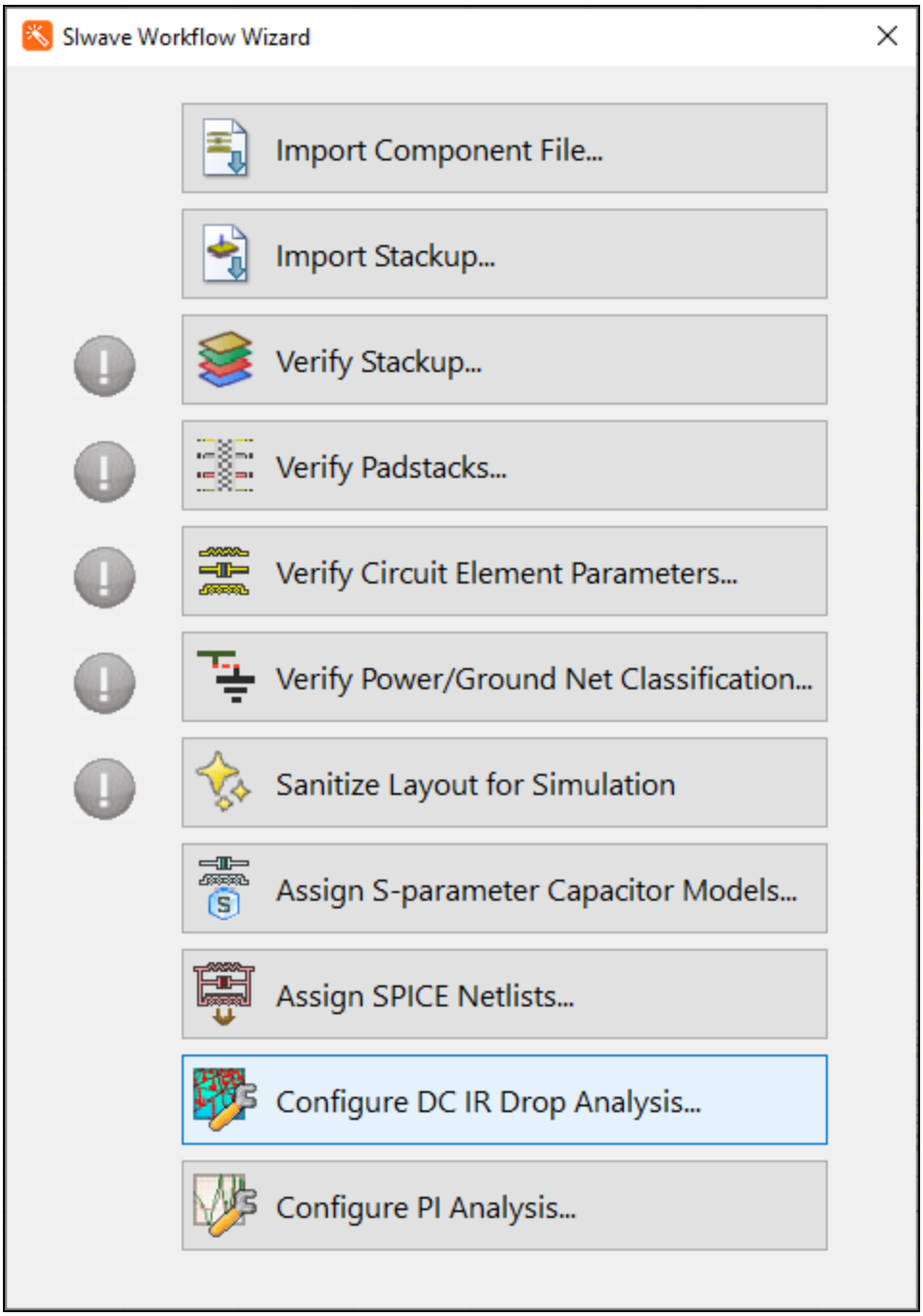
To complete these solutions from General mode, begin with the SIwave Workflow Wizard, then follow the specific steps for the solver.

SIwave Workflow Wizard in HFSS 3D Layout (General Mode Only)

The SIwave Workflow Wizard walks you through the steps to make a project ready for SIwave simulations. Follow these steps to access the wizard.

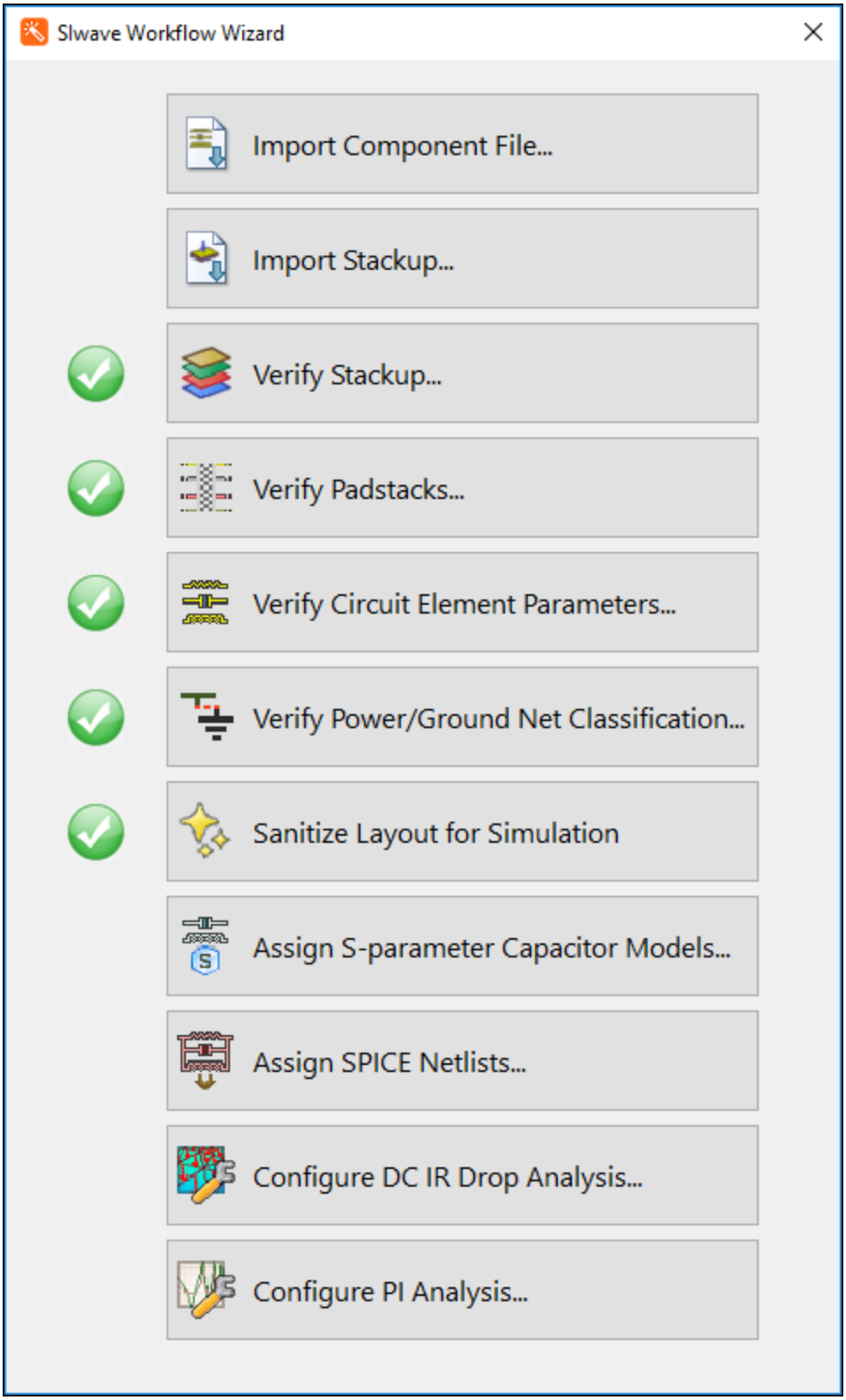
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Layout** tab, click **Workflow** to open the **SIwave Workflow Wizard** window.



2. Steps with an exclamation point next to them should be completed before running any SIwave simulations in HFSS 3D Layout. Other steps may be appropriate, depending on the simulation type. Once completed, the exclamation points turn into green check marks.

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.

SIwave Workflow Wizard steps include:

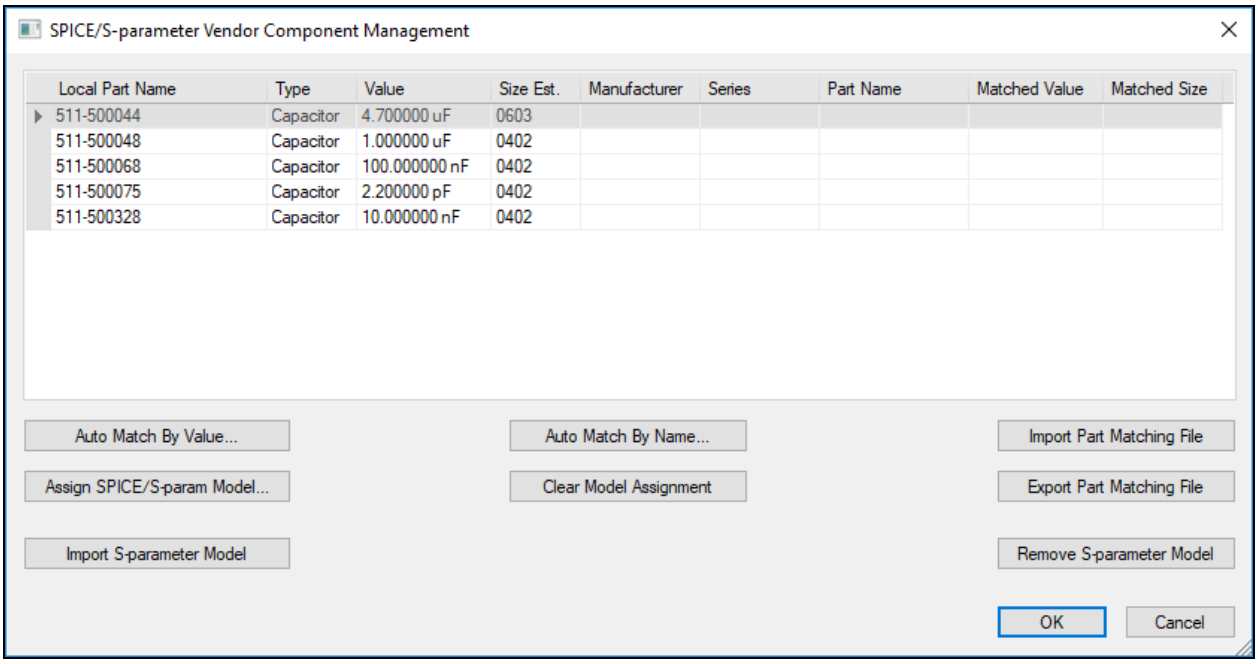
- **Import Component File** – allows you to import component files (*.cmp) generated by third-party layout tools so you do not have to re-draw components while transferring the design. These files contain detailed pin information for discrete devices, input/output devices and integrated circuits.
- **Import Stackup** – allows you to import a stackup from an Extensible Markup Language (*.xml) file or layer stackup file (*.stk). PCB/package designs are often based on the same stackup. Stackup file import/export allows you to define the exact parameters (set materials, layer thicknesses, etc.) for a particular stackup once, and have these assignments be reapplied consistently across different designs.
- **Verify Stackup** – opens the **Edit Layers** window / **Layer Stackup Wizard** so that you can verify the layers and materials used in the design.
- **Verify Padstacks** – opens the **Padstack Instances** window, where you can verify or edit padstacks.
- **Verify Circuit Element Parameters** – opens the **DC Simulation Element Data** window, where you can verify settings for Capacitors, Inductors, Resistors, Ports, Current Sources, Voltage Sources, and Net Terminals.
- **Verify Power/Ground Net Classification** – allows you to specify Power/Ground Nets and Non Power/Ground Nets.
- **Sanitize Layout for Simulation** – performs an operation for cleaning power and ground nets in order to fix certain alignment problems and complexities that may slow down simulation. This works by uniting the planes and traces for each net to be cleaned. Once the united planes are formed, portions that display trace-like properties are converted into traces. Under certain conditions, these traces are extended in order to improve their connection with vias, pads, and the remaining planes. Lattices originally formed from multiple traces become planes with cutouts after the sanitization process.
- **Assign S-parameter Capacitor Models** – opens the **SPICE/S-parameter Vendor Component Management** window, where you can assign SPICE/S-parameter models to components.
- **Assign SPICE Netlists** – opens the **RLC SPICE Model Setup** window, where you can attach SPICE models to components.
- **Configure DC IR Drop Analysis** – opens the **DC IR Configuration** window, a step in running DC IR simulations.
- **Configure PI Analysis** – opens the **PI Configuration** window for running the Ansys SIwave™ SYZ Solver.

Assigning SPICE/S-parameter Vendor Models in HFSS 3D Layout

To assign SPICE/S-parameter models to components in HFSS 3D Layout:

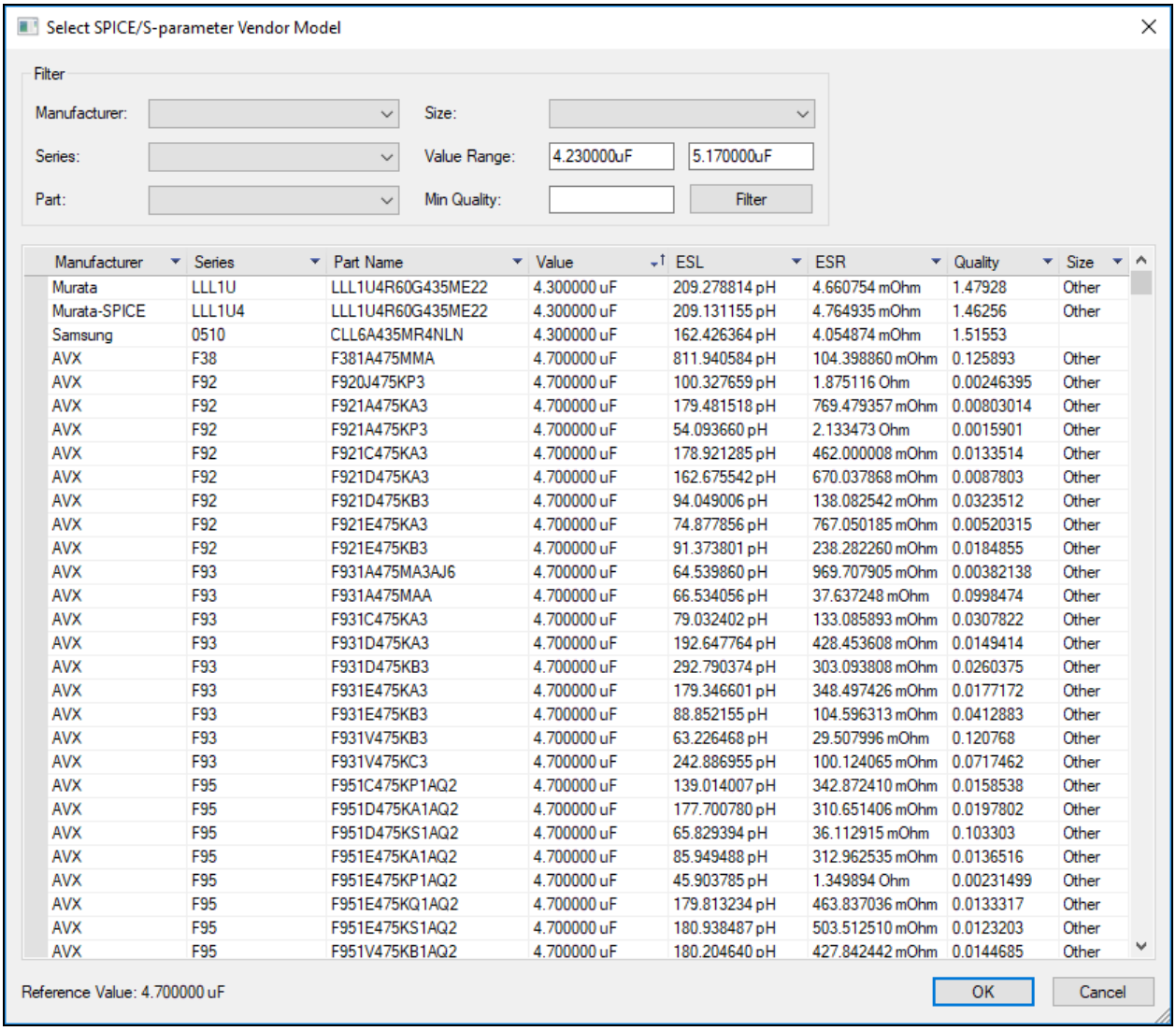
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Layout** tab, click **Workflow** to open the SIwave Workflow Wizard.
2. Click **Assign S-Parameter Capacitor Models** to open the **SPICE/S-parameter Vendor Component Management** window.



3. Select a component from the list.
4. If you have an SIwave Partname Mapping File (*.pmap), click **Import Part Matching File** to load those settings.
5. Assign specific models to parts automatically by clicking **Auto Match By Value** or **Auto Match By Name**.
6. To assign a model to a part manually, click **Assign SPICE/S-param Model** to open the **Select SPICE/S-parameter Vendor Model** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

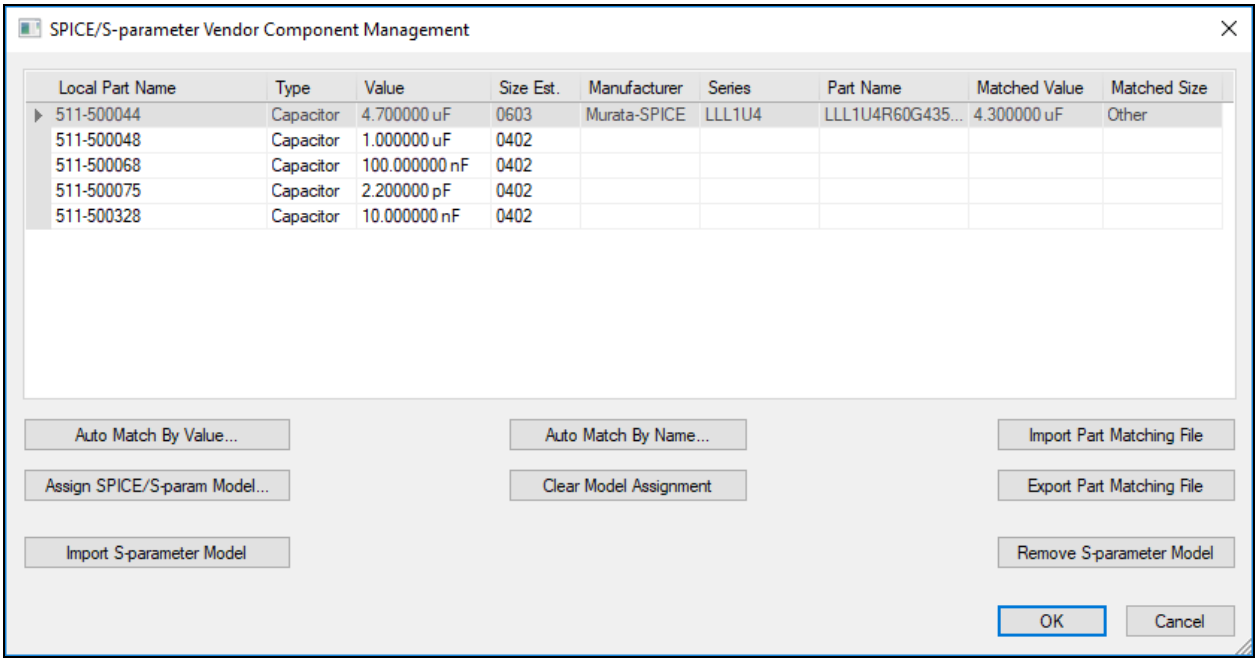


Manufacturer	Series	Part Name	Value	ESL	ESR	Quality	Size
Murata	LLL1U	LLL1U4R60G435ME22	4.300000 uF	209.278814 pH	4.660754 mOhm	1.47928	Other
Murata-SPICE	LLL1U4	LLL1U4R60G435ME22	4.300000 uF	209.131155 pH	4.764935 mOhm	1.46256	Other
Samsung	0510	CLL6A435MR4NLN	4.300000 uF	162.426364 pH	4.054874 mOhm	1.51553	
AVX	F38	F381A475MMA	4.700000 uF	811.940584 pH	104.398860 mOhm	0.125893	Other
AVX	F92	F920J475KP3	4.700000 uF	100.327659 pH	1.875116 Ohm	0.00246395	Other
AVX	F92	F921A475KA3	4.700000 uF	179.481518 pH	769.479357 mOhm	0.00803014	Other
AVX	F92	F921A475KP3	4.700000 uF	54.093660 pH	2.133473 Ohm	0.0015901	Other
AVX	F92	F921C475KA3	4.700000 uF	178.921285 pH	462.000008 mOhm	0.0133514	Other
AVX	F92	F921D475KA3	4.700000 uF	162.675542 pH	670.037868 mOhm	0.0087803	Other
AVX	F92	F921D475KB3	4.700000 uF	94.049006 pH	138.082542 mOhm	0.0323512	Other
AVX	F92	F921E475KA3	4.700000 uF	74.877856 pH	767.050185 mOhm	0.00520315	Other
AVX	F92	F921E475KB3	4.700000 uF	91.373801 pH	238.282260 mOhm	0.0184855	Other
AVX	F93	F931A475MA3AJ6	4.700000 uF	64.539860 pH	969.707905 mOhm	0.00382138	Other
AVX	F93	F931A475MAA	4.700000 uF	66.534056 pH	37.637248 mOhm	0.0998474	Other
AVX	F93	F931C475KA3	4.700000 uF	79.032402 pH	133.085893 mOhm	0.0307822	Other
AVX	F93	F931D475KA3	4.700000 uF	192.647764 pH	428.453608 mOhm	0.0149414	Other
AVX	F93	F931D475KB3	4.700000 uF	292.790374 pH	303.093808 mOhm	0.0260375	Other
AVX	F93	F931E475KA3	4.700000 uF	179.346601 pH	348.497426 mOhm	0.0177172	Other
AVX	F93	F931E475KB3	4.700000 uF	88.852155 pH	104.596313 mOhm	0.0412883	Other
AVX	F93	F931V475KB3	4.700000 uF	63.226468 pH	29.507996 mOhm	0.120768	Other
AVX	F93	F931V475KC3	4.700000 uF	242.886955 pH	100.124065 mOhm	0.0717462	Other
AVX	F95	F951C475KP1AQ2	4.700000 uF	139.014007 pH	342.872410 mOhm	0.0158538	Other
AVX	F95	F951D475KA1AQ2	4.700000 uF	177.700780 pH	310.651406 mOhm	0.0197802	Other
AVX	F95	F951D475KS1AQ2	4.700000 uF	65.829394 pH	36.112915 mOhm	0.103303	Other
AVX	F95	F951E475KA1AQ2	4.700000 uF	85.949488 pH	312.962535 mOhm	0.0136516	Other
AVX	F95	F951E475KP1AQ2	4.700000 uF	45.903785 pH	1.349894 Ohm	0.00231499	Other
AVX	F95	F951E475KQ1AQ2	4.700000 uF	179.813234 pH	463.837036 mOhm	0.0133317	Other
AVX	F95	F951E475KS1AQ2	4.700000 uF	180.938487 pH	503.512510 mOhm	0.0123203	Other
AVX	F95	F951V475KB1AQ2	4.700000 uF	180.204640 pH	427.842442 mOhm	0.0144685	Other

7. Click the **Filters** or scroll through the list to select a model.
8. Click **OK** to return to the **SPICE/S-parameter Vendor Component Management** window.

   The new model assignment displays in the table.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



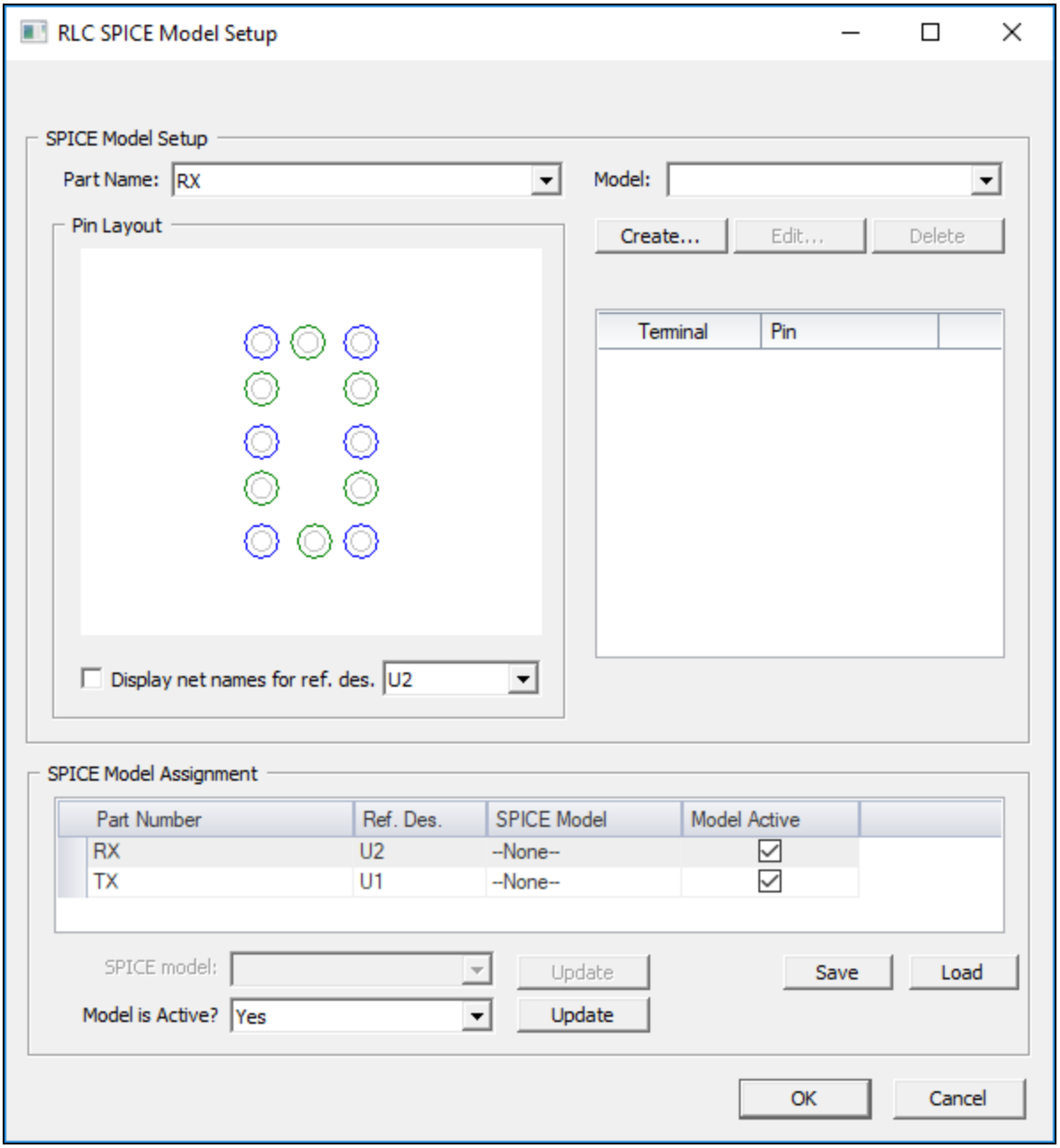
9. If applicable, you can clear a model assignment by selecting the part and clicking **Clear Model Assignment**.
10. If selected, click **Export Part Matching File** to export your settings as an SIwave Partname Mapping File (*.pmap).
11. Click **OK** to save the settings and return to the SIwave Workflow Wizard.

Attaching a SPICE RLC Model to a Component in HFSS 3D Layout

You can attach a SPICE RLC model to a specific component in the layout or to a component type in multiple ways. This model can include an arbitrary Netlist when the component's Solve Independent option is enabled (i.e., selected for inclusion by Linear Network Analysis only).

1. From the SIwave Workflow Wizard:
    a. From the **Layout** tab, click **Workflow** to open the wizard.
    b. Click **Assign SPICE Netlists**.
2. From the **Properties** window:
    a. From the **Layout Editor**, click a component.
    b. From the **Properties** window, click the button in the **Model Info** row.
    c. From the **Component Model** window, click **SPICE model** on the **Type** drop-down menu.
    d. Click **Import SPICE Model** to open the **RLC SPICE Model Setup** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

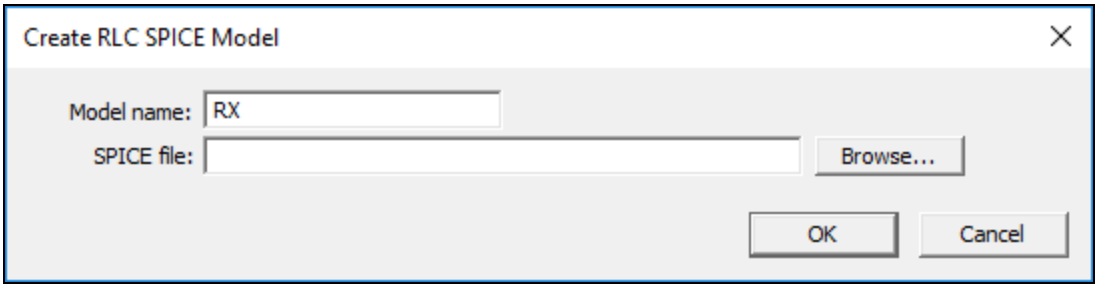


3. Select the **Part Name** to display its **Pin Layout**.
4. To see the net assignments for the component's pins, check the **Display net names for ref. des.** check box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Select the **Model**:
    a. Choose a model from the drop-down menu, or click **Create** to create a new one.

       The **Create RLC SPICE Model** window opens, showing the same **Model name** as the **Part Name**.



    b. Type the name and location of a SPICE netlist, or
    c. Click **Browse** to locate a file you want to attach to the selected part.

       The model name is now set to the SPICE subcircuit name, but can be edited.

6. Click **OK** to close the **Create RLC SPICE Model** window

   If coming on the **Component Model** window, you are done.

   If coming on the **SIwave Workflow Wizard**, the selected model is loaded and the terminals in the SPICE subcircuit display. Continue as follows.

7. Map **Terminals** in the SPICE subcircuit to corresponding **Pins** belonging to the part.

> **Note:**
>
> Mapping must be performed correctly, as simulation results depend on proper connections between part pins and the SPICE subcircuit terminals.

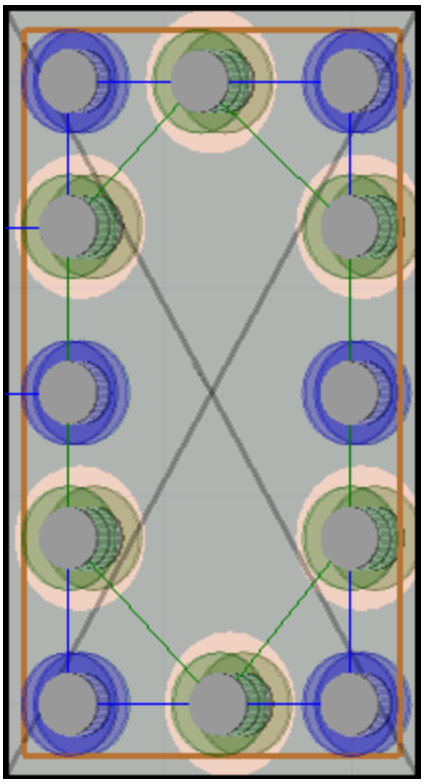
8. Assign the SPICE model, in the **SPICE Model Assignment** area:
    a. Select a **Part Number** and **Reference Designator**.
    b. Make a selection on the **SPICE Model** drop-down menu.
    c. Select the **Model Active** check box if you want to assign a model and want it to be active. If cleared, the model is assigned but is not active.
9. If you want to assign the same SPICE model and model active settings to multiple parts:
    a. Select the reference designators to which you want to assign the same SPICE model. You can select multiple reference designators in the table using **Ctrl** and **Shift**, or select all using **Ctrl** +**A**.
    b. Select the **SPICE model** on the dropdown menu.
    c. Click **Update**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. To activate the model for all the selected parts, select **Yes** on the **Model is Active?** drop-down menu, and click **Update**.

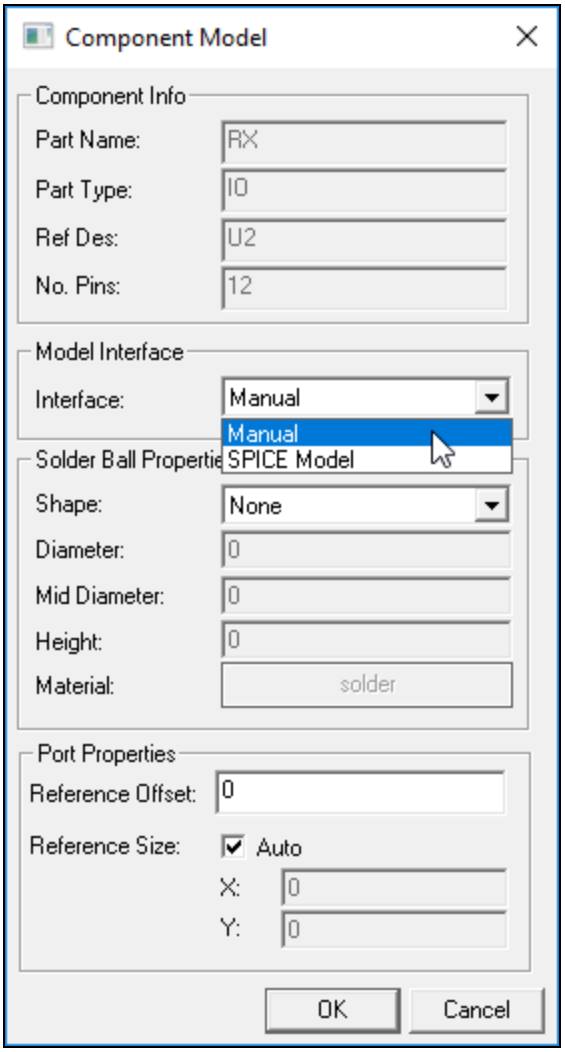
10. Click **Save** to save the model and open the **Select Spice Model Assignment Output File Name** window. Select the location to store the saved file. Enter the name for the file in the **File name** field. Select the file type on the **Save as type** area. You can save files with .smas (Spice Model Assignment File) extensions. Click **Save**.
11. Click **Load** to load the saved values from a file and open the **Select Spice Model Assignment File to load** window. Select the file you want to open. You can open saved .smas files. Click **Open**.
12. To confirm the SPICE model assignments and return to the SIwave Workflow Wizard, click **OK**.

To indicate the connection of a SPICE model, a gray rectangle containing an **X** is displayed over the reference designator(s) in the Modeling workspace:



To activate or deactivate the SPICE model in the **Properties** window, click **Model Info** and set the Component Model **Interface** to **SPICE Model**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

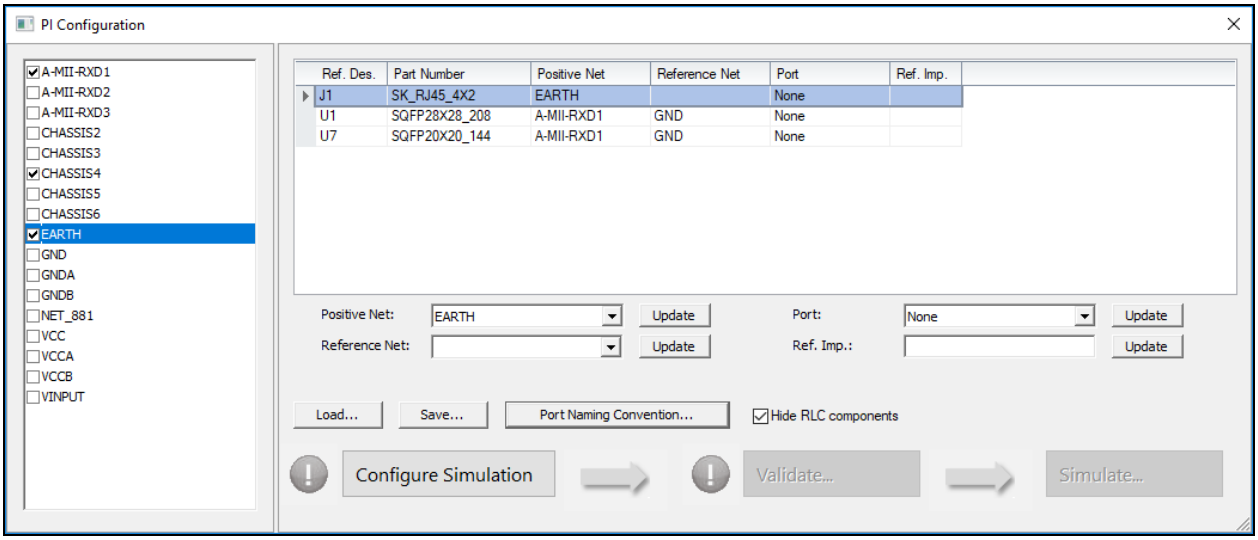


Computing SYZ Solutions in HFSS 3D Layout (General Mode Only)

Before starting SIwave SYZ extraction, you must define ports at the connection points for the sources and measurements, and identify the power and ground nets in your design. This is done via the SIwave Workflow Wizard's **PI Configuration** window.

> **Note:**
>
> To complete a simulation of ECAD-ECAD hierarchy (e.g., packages mounted on a PCB) or MCAD 3D components using HFSS region-based simulations, refer to SIwave Region-Based Simulations in HFSS 3D Layout.

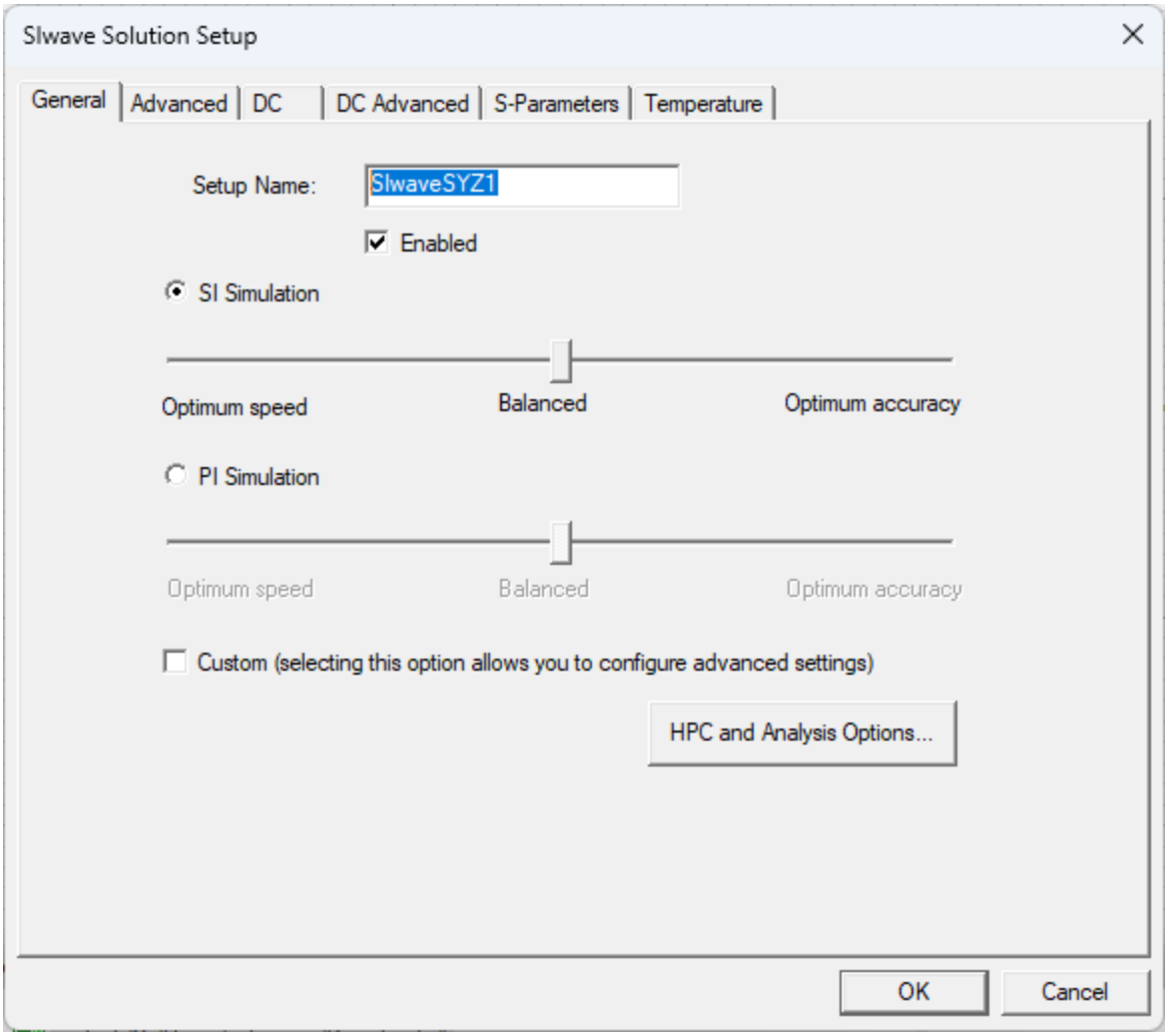
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To Compute the S-,Y-, and Z-Parameters

1. From the **Project Manager** window, right-click **Analysis** and select **Add SIwave Solution Setup** to open the **SIwave Solution Setup** window opens, on the **General** 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. Select either **SI simulation** (Signal Integrity) or **PI simulation** (Power Integrity), and move the corresponding slider to adjust speed and accuracy.

   For SI simulations:

   - **Optimum speed** – of the Coupling options on the **Advanced** tab, only Trace Coupling are selected.
   - **Balanced** – of the Coupling options on the **Advanced** tab, Trace, Coplane, Split-Plane and Cavity Field Coupling are selected.
   - **Optimum accuracy** – Trace Return Current Distribution are selected, in addition to the Balanced Coupling options.

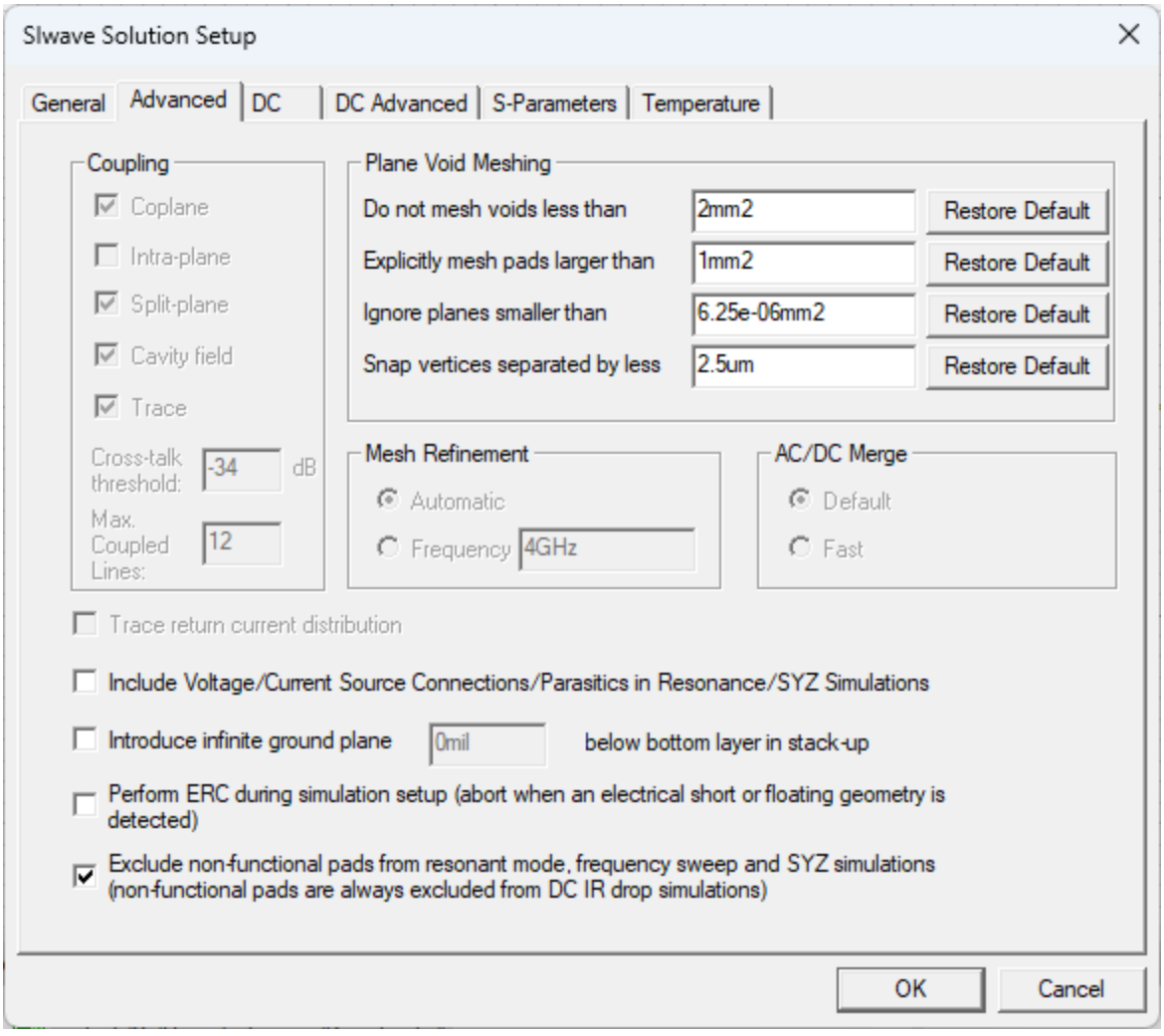
   For PI simulations:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Optimum speed** – no coupling options are selected.
- **Balanced** – Cavity Field coupling are selected.
- **Optimum accuracy** – Trace, Coplane, and Split-Plane Coupling are selected, in addition to Cavity Field.

The **Custom** check box allows you to configure additional options on the **Advanced** tab.

3. Click **HPC and Analysis Options** to open a window that allows you to set HPC options.
4. Click the **Advanced** tab.



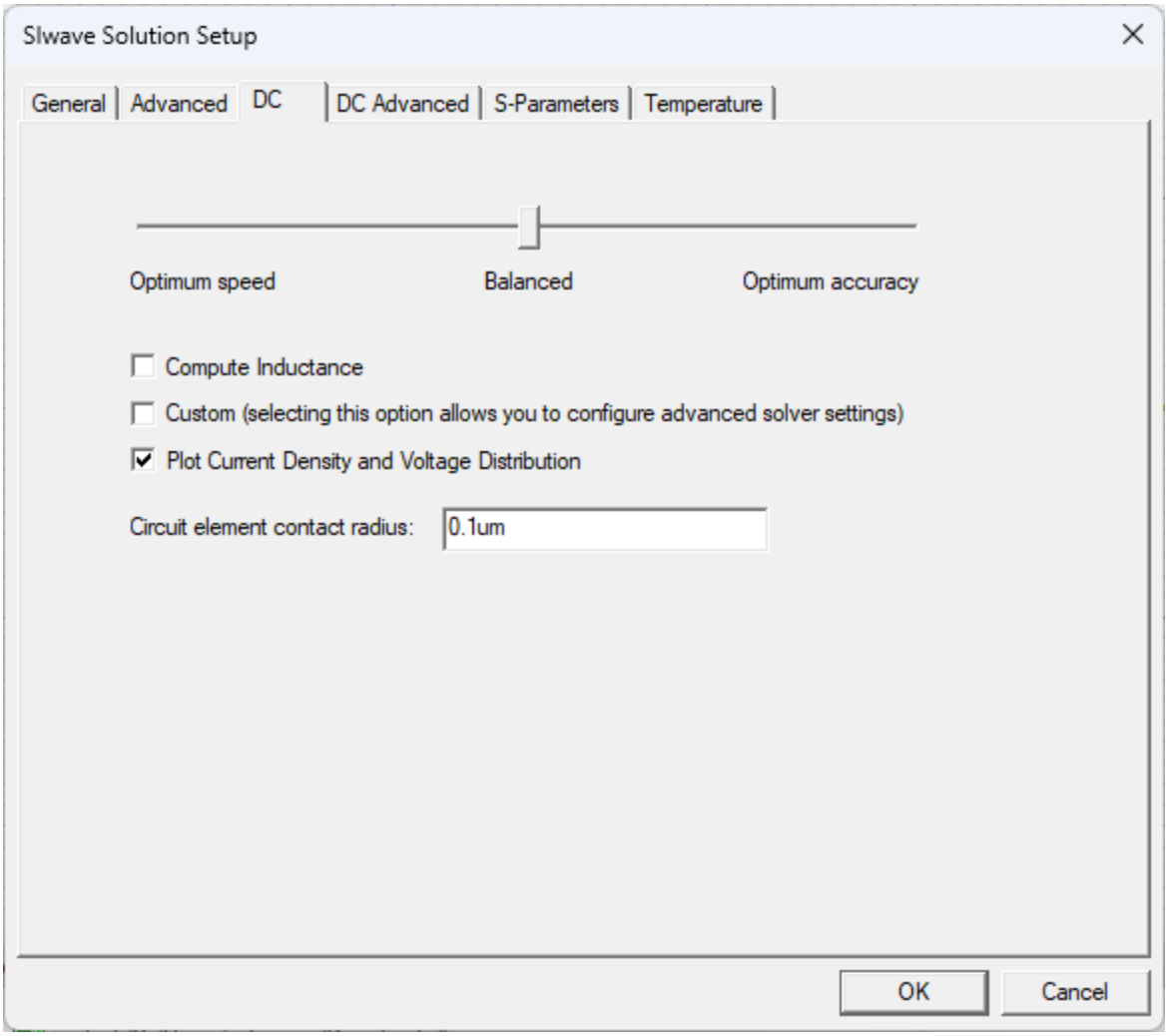
5. Set the **Advanced** tab options:
   - **Coupling** – selections on the **General** tab display and are locked. If you chose **Custom** on the General tab, select on the listed options.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Plane Void Meshing** – specify plane void meshing, control geometry defeaturing, and vertex snapping measurements and units, or **Restore Default** settings.
- **Mesh Refinement** – either select **Automatic** or specify a **Frequency**.
- **Trace return current distribution** – available only if you have selected the **Custom** check box in the **General** tab. Check this option to accurately model the change of the characteristic impedance of transmission lines caused by a discontinuous ground plane. Instead of injecting the return current of a trace into a single point on the ground plane, the return current for a high impedance trace is spread out. The trace return current is not distributed when all traces attached to a node have a characteristic impedance of less than 75 ohms, or if the difference between two connected traces is less than 25 ohms.
- **Include Voltage/Current Source Connections/Parasitics in Resonance/SYZ Simulations** – determine whether to include the effect of parasitic elements from voltage and current sources.
- **Introduce infinite ground plane** – specify the elevation of the infinite unconnected ground plane to place under the design. The ground plane serves as a voltage reference for traces and planes if they are not defined in the layout.
- **Perform ERC during simulation setup** – determine whether to perform error checking while generating the solver input file. In some designs, the same net may be divided into multiple nets with separate names; these nets are connected at a “star point.” To simulate these nets, error checking for DC shorts must be deactivated. All overlapping nets are internally united during simulation.
- **Exclude non-functional pads from resonant mode, frequency sweep and SYZ simulations** – determine whether to remove non-functional pads. All layers in which a via does not make an electrical connection to any other geometry do not contain a pad. By default, non-functional pads are not included in the simulation. Typically non-functional pads are removed when a PCB is manufactured.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Select the **DC** tab.

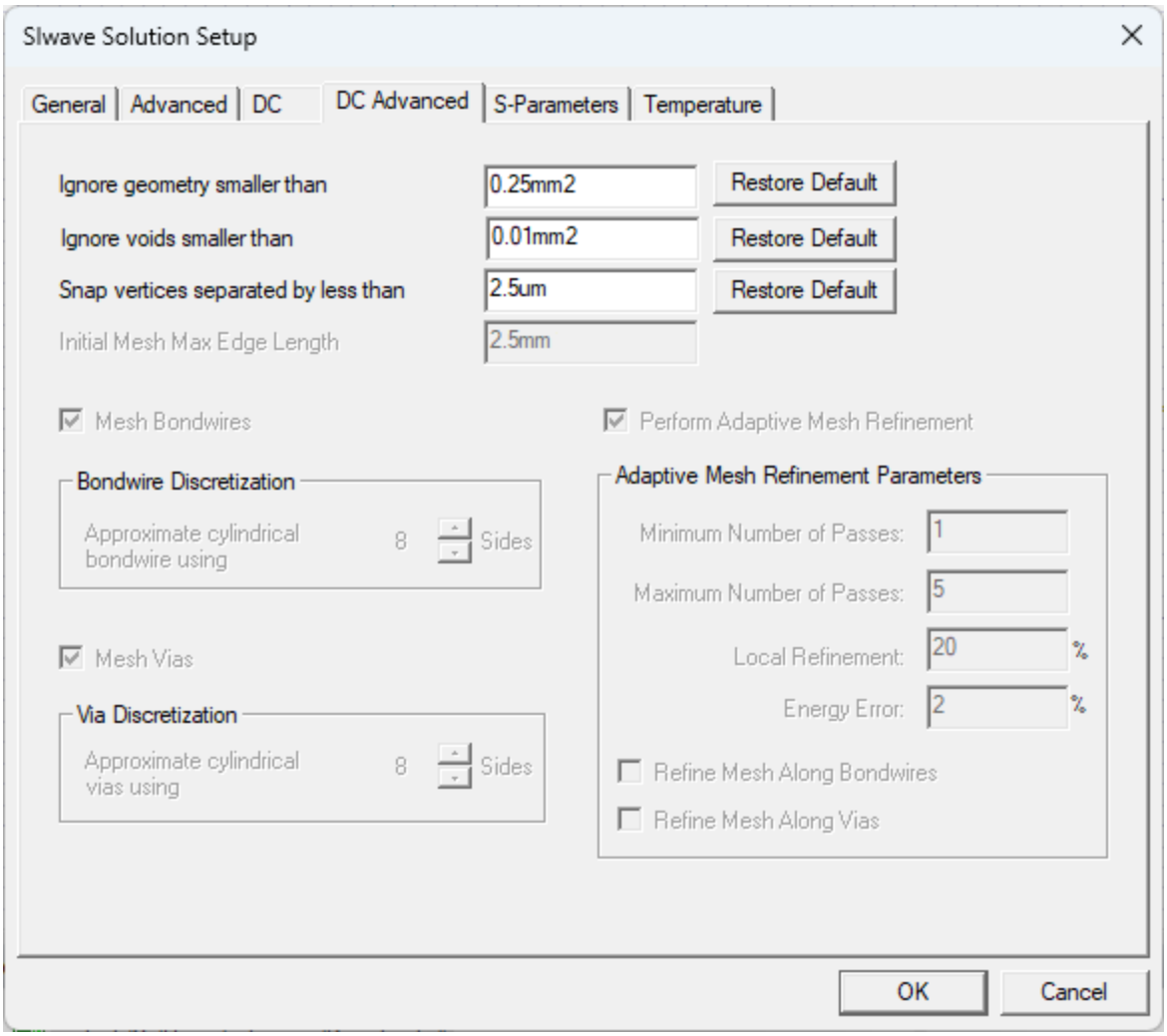


7. Set the following options:
   - Use the slider to balance speed and accuracy:
     - **Optimum Speed** – all options in the **DC Advanced** tab are deactivated.
     - **Balanced** – the Mesh Bondwires and Mesh Vias options are selected, along with adaptive Mesh Refinement.
     - **Optimum Accuracy** – the mesh are refined on bondwire and vias, and more aggressive refinement parameters are specified.
   - Select whether to **Compute Inductance**. The inductance is computed by integrating the vector potential at DC using an efficient, multi-level fast multipole method with two-level parallelization. In some cases, this may require significant computational time and memory.
   - Select whether to use **Custom** simulation settings.

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 **Plot Current Density and Voltage Distribution**. Clear this check box to save disk space if you don't need to post-process voltage and current plots (e.g., if you only want to export a SPICE netlist after a DC simulation, it might not be helpful to save any field data to disk).
- Enter the **Circuit Element Contact Radius**. Equal potential is enforced in the contact radius, and no mesh refinement is done inside the contact region.

8. Select the **DC Advanced** tab.



DC Advanced options include:

- **Ignore geometry smaller than** – geometry whose area (after being combined with any overlapping geometry) falls under this specified threshold is ignored during simulation.
- **Ignore voids smaller than** – geometry whose area (after being combined with any overlapping geometry) falls under this specified threshold is ignored during simulation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

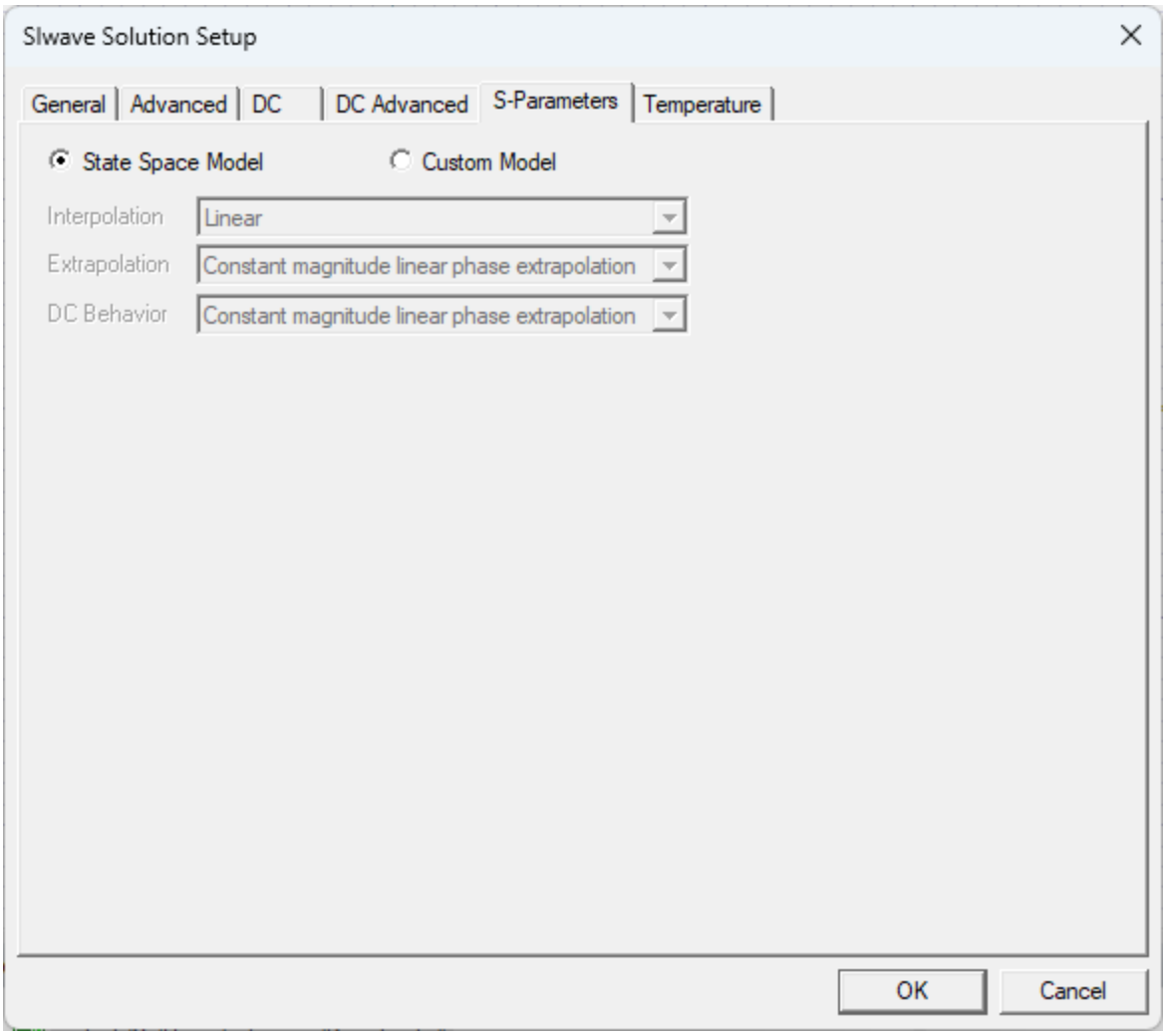
- **Snap vertices separated by less than** – entities (e.g., vias) and vertices that are separated by less than the distance specified in this box may be snapped together during solver pre-processing.
- **Initial Mesh Max. Edge Length** – specifies the maximum edge length selected in the mesh (before any adaptive refinement occurs).
- **Mesh Bondwires** – check if users do not want to model bondwires as point connections to mesh. Specify the **Bondwire Discretization** (the number of sides used to approximate cylindrical bondwires). Specify any value between 6 and 64. The default value is 8.
- **Mesh Vias** – check if users do not want to model vias as point connections to mesh. Specify the **Via Discretization** (the number of sides used to approximate cylindrical vias). Specify any value between 6 and 64. The default value is 8.
- **Perform Adaptive Mesh Refinement** – check to perform Adaptive Mesh Refinement, and populate the fields:
    - **Minimum Number of Passes** – an adaptive analysis does not stop unless the minimum number of passes specified has been completed, even if convergence criteria have been met.
    - **Maximum Number of Passes** – this value is the maximum number of mesh refinement cycles users want the simulation to perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops; otherwise, the adaptive analysis continues unless the convergence criteria are reached.

      > **Note:**
      >
      > The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform requesting more memory than is available or taking excessive time to compute solutions.

    - **Local Refinement** – this value determines how many triangles are added at each iteration of the adaptive refinement process. The triangles with the highest error is refined. The default value is 20%.
    - **Energy Error** – the default value is 2%.
- **Refine Mesh Along Bondwires** – check to refine mesh along bondwires.
- **Refine Mesh Along Vias** – check to refine mesh along vias.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Select the **S-Parameters** tab.



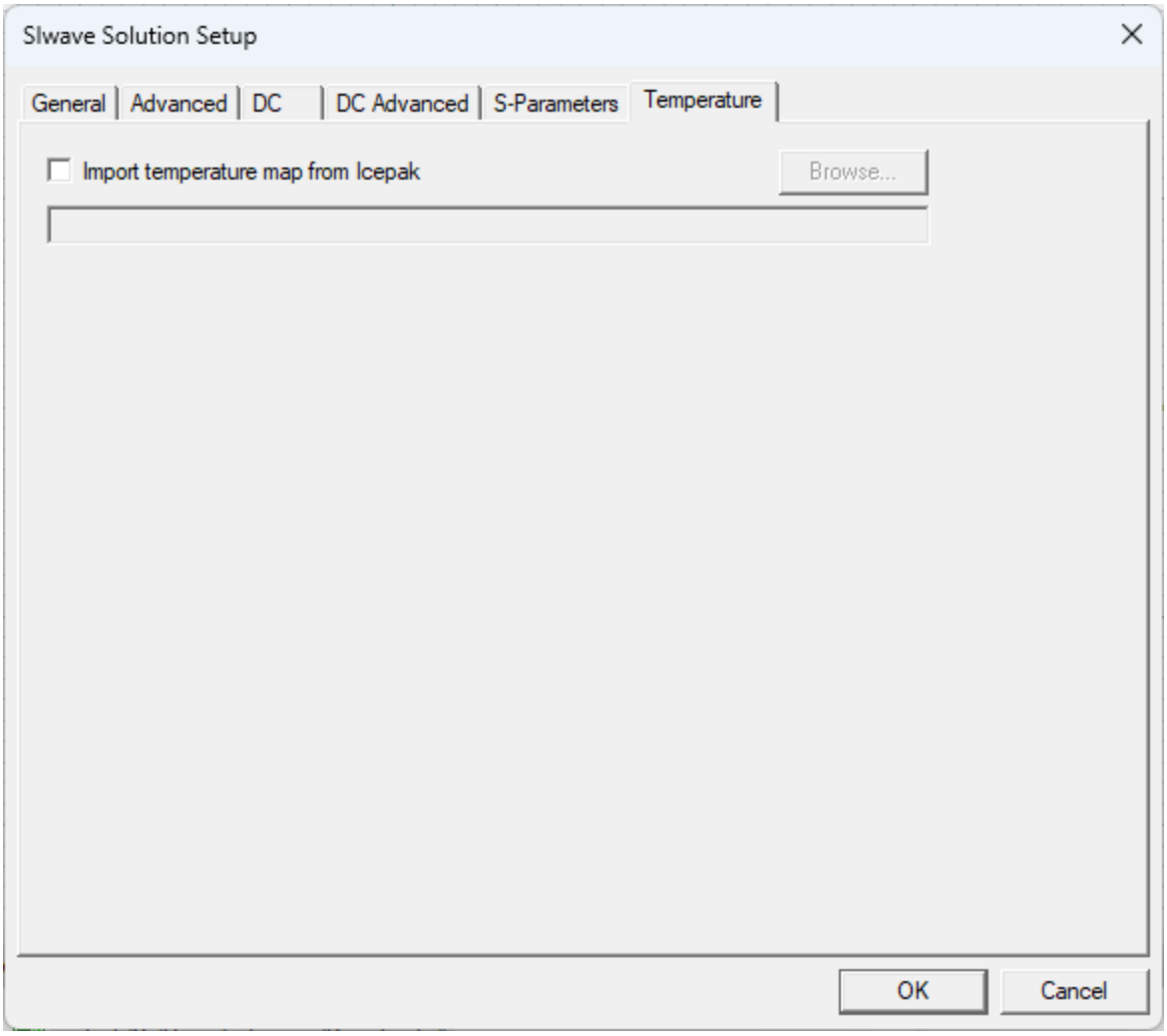
10. Select **State Space Model** or **Custom Model**. If **Custom Model** is selected, set the following options:
    - **Interpolation** – select **Point**, **Linear**, or **Step** from the drop-down menu.
    - **Extrapolation** – select **Zero padding**, **Same as last point**, **Linear extrapolation from last 2 points**, or **Constant magnitude linear phase extrapolation** from the drop-down menu.
    - **DC Behavior** – select **Zero padding**, **Same as last point**, **Linear extrapolation from last 2 points**, or **Constant magnitude linear phase extrapolation**, **Series Capacitor**, or **Leave all signal lines open circuited** 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.

Note:

Refer to the following pages for an example of using the **SIwave Solution Setup** window to change S-Parameters or using a preconfigured Python script to generate an Ansys ECAD database file, which can then be imported into AEDT to create a custom SIwave setup.

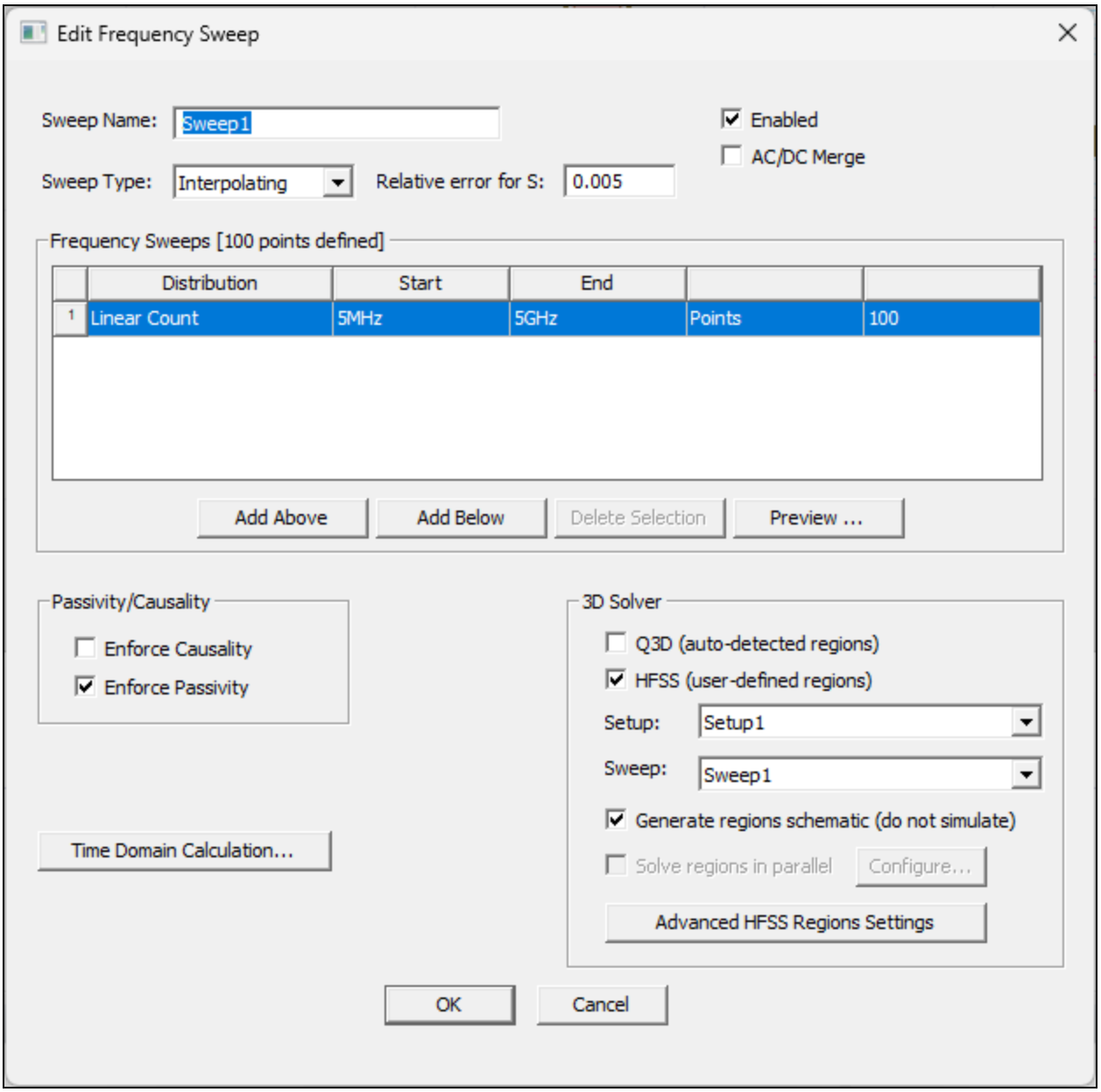
11. Select the **Temperature** tab.



12. If necessary, check **Import temperature map from Icepak** to select either an Icepak simulation's results, or an external Ansys Icepak/Ansys SIwave temperature distribution file, then **Browse** to an appropriate file and select it (e.g., test.sitemp). The temperature data can locally adjust DC conductivity.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Click **OK** to open the **Edit Frequency Sweep** window.



Note:

Access this window on the **Project Manager** window. Expand the SIwave solution and double-click the Frequency Sweep.

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 following options:
  - Select the type of sweep on the Sweep Type drop-down menu. Select **Interpolating** (default), **Discrete** or **Broadband Fast**. If you select the **Interpolating** or **Broadband Fast** sweep types, you must also specify the error tolerance (**Relative error for S**). An interpolating sweep estimates a frequency response for an entire range by solving a relatively small number of frequency points in the range and interpolating the remaining points.
  - Choose whether to check the **Enhanced AC/DC merge** box to enforce causality and generate S-parameters at the DC point. By combining the DC results with the frequency-swept AC results, the simulation will have improved improve accuracy over the entire bandwidth.

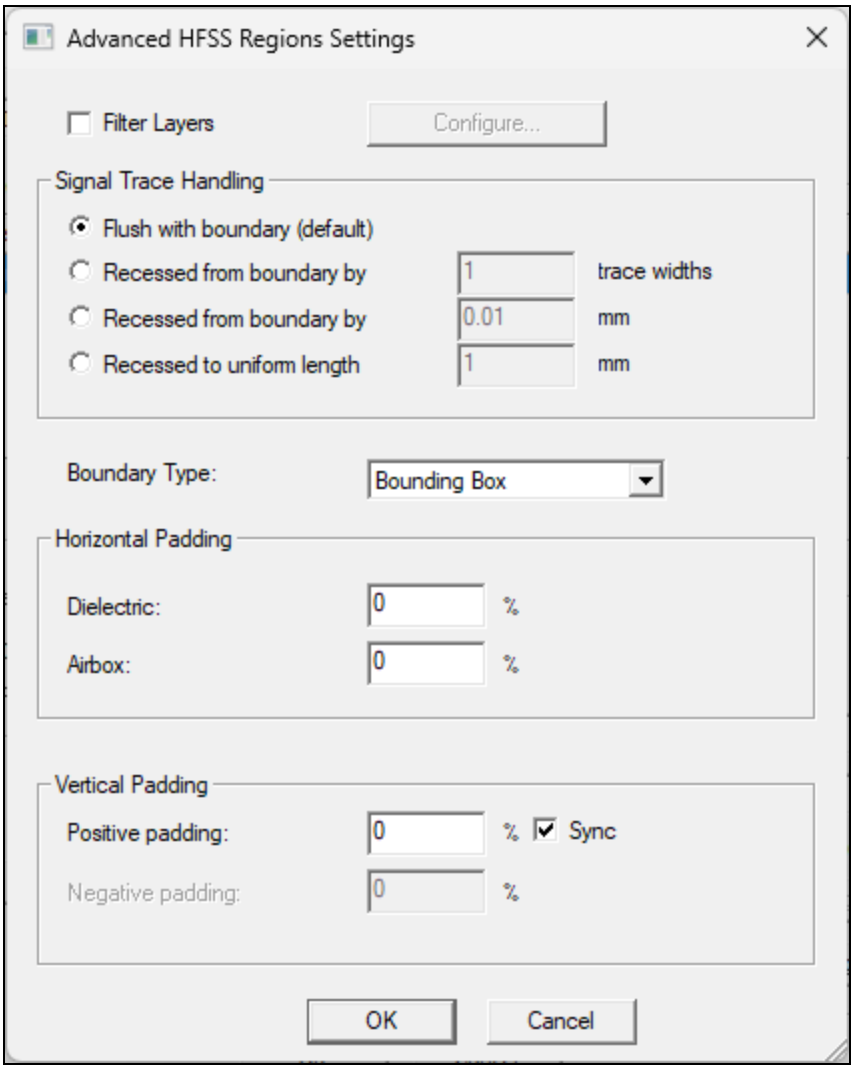
> **Note:**
>
> The following guidelines must be followed:
>
> - You need to manually define regions. Regions must contain signal net geometry that is included in the simulation. A region containing only power/ground net geometry or signal net geometry that is not included in the simulation is invalid.
> - Ports must exist either completely inside or completely outside of a region. Simulation configurations consisting of ports that straddle regions are invalid.
> - HFSS regions (which are user-defined) and Q3D regions (determined by the solver) can coexist.
> - Power/ground nets should be properly classified.
> - Signal net geometry in the region, along with power/ground net geometry, is solved using HFSS and the resulting S-parameters are incorporated onto the SIwave solve to compute a unified channel response.

14. If present, use the check boxes in the **Passivity/Causality** area to enforce causality and/or passivity.
15. From the **3D Solver** area, check the **Q3D (auto-detected regions)** box to utilize the Q3D solvers for 3D-type regions such as unreferenced traces, complex via transitions and pad coupling.
16. From the **3D Solver pane**, check the **HFSS (user-defined regions)** box to use the predefined HFSS regions and launch the HFSS simulation options. Selecting HFSS enables options to generate a regions schematic and solve regions in parallel. These options are mutually exclusive.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> HFSS regions must include signal net geometry that is part of the simulation. Regions containing only power/ground nets or excluded signal nets are invalid. HFSS and Q3D regions can coexist, but ports must not straddle region boundaries.

17. If appropriate, click **Advanced HFSS Regions Settings** to open the **Advanced HFSS Regions Settings** window.



Use the **Advanced HFSS Regions Settings** window to configure simulation boundaries and trace behavior for user-defined HFSS regions in SIwave region-based simulations. These settings affect how dielectric and airbox extents are clipped, how trace ends are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

recessed, and how vertical padding is applied. Refer to Ansys SIwave™ Region-Based Simulations in HFSS 3D Layout.

a. To enable layer filtering, check the **Filter Layers** box. Then click **Configure** to open a **Layer Selection** window. Manually choose which signal layers are included in the HFSS region definition by checking adjacent boxes. Only selected layers will be included when defining HFSS regions. Click **OK** to confirm selection and close the **Layer Selection** window.

> **Note:**
>
> Filtering layers helps exclude irrelevant geometry and ensures that only signal nets intended for simulation are included in the HFSS region. Unselected layers will be excluded from simulation, and all geometry on those layers will be deleted. If all layers containing relevant signal geometry are not included, path discontinuities may result. Path discontinuity may also result if relevant GND layers are excluded (typically those immediately above and below signal layers). For example, if signal geometry exists on the Top, Bottom, and Layer 7, you should also include Layer 1 (below Top), Layer 2 (above Bottom), and Layers 6 and 8 (adjacent to Layer 7). The full set would be: Layer 1, Layer 2, Layer 6, Layer 7, Layer 8, Layer N-1, Layer N, et cetera.

b. From the **Signal Trace Handling** area, choose how signal traces are recessed from the region boundary:
   - **Flush with boundary (default)**: Virtual ports are placed directly on the region boundary.
   - **Recessed from boundary by**: Traces are clipped to create a gap between the virtual ports and the boundary. The recess is specified as a multiple of trace widths.
   - **Recessed from boundary by**: Traces are clipped to create a gap between the virtual ports and the boundary. The recess is specified in millimeters.
   - **Recessed to uniform length**: Traces are clipped so the remaining trace length is uniform across all traces in millimeters.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Virtual ports must lie entirely within the region to be simulated. In the case of virtual ports generated flush to the boundary, trace routes are modified and a generous airbox is required to meet this constraint. For simulations with tight, conformal dielectric and airboxes, virtual ports must be recessed from the boundary to meet this constraint.

c. From the **Boundary Type** drop-down menu, select the clipping method for both dielectric and airbox boundaries:
   - **Bounding Box**: Adds horizontal and vertical padding beyond the geometry.
   - **Conformal**: Clips exactly to the board outline (i.e., 0% padding).

d. From the **Horizontal Padding** area, enter values for:
   - **Dielectric**: Percentage of horizontal padding added around the dielectric region.
   - **Airbox**: Percentage of horizontal padding added around the airbox.

   > **Note:**
   >
   > If **Flushed with boundary (default)** is selected, the boundary type is forcibly set to **Bounding Box**. A generous horizontal and vertical padding of approximately **15–25%** is recommended. If **Recessed from boundary by** or **Recessed to uniform length** is selected, the boundary type is forcibly set to **Conformal**.

e. From the **Vertical Padding** area:
   - Enter values for **Positive padding** and **Negative padding**, as percentages.
   - Check the **Sync** box to apply the same value to both directions.

   > **Note:**
   >
   > Vertical padding defines the extent of the simulation region above and below the board. Excessive padding may increase simulation time.

f. Click **OK** to apply the settings or **Cancel** to discard changes.

18. Click **OK** to complete the solution setup.
19. From the **Project Manager** window, right-click the solution and select **Analyze**. The **Progress** window shows simulation progress. When the simulation has finished, view and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

export results.

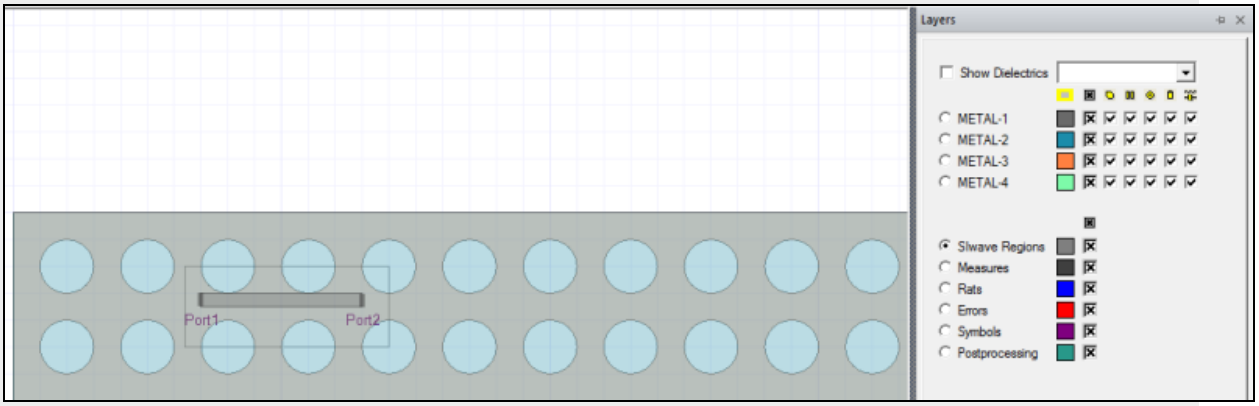
Ansys SIwave™ Region-Based Simulations in HFSS 3D Layout

Simulating package/PCB layouts using Ansys SIwave™ printed circuit board (PCB) and package electromagnetics simulation software with Ansys HFSS™ regions defined around 3D structures (e.g., bondwires, irregular via transitions, BGA breakouts, et cetera), including ECAD-ECAD hierarchy (e.g., packages mounted on a PCB) and MCAD 3D components, can provide an optimal balance of accuracy and simulation time.

In a standard Ansys HFSS simulation, users designate areas in a layout as Ansys HFSS regions. During simulation, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform automatically segments the selected regions into separate projects, generates virtual ports on relevant signal nets, and solves the selected regions sequentially in HFSS. Electronics Desktop then solves any geometry outside of those Ansys HFSS Regions with Ansys SIwave. As long as the points of contact (e.g., solder balls) between EDB cells lie within the selected HFSS regions, AEDT maintains simulation speed without sacrificing accuracy due to design segmentation (Refer to Hierarchical Restrictions). The resulting S-matrices from both Ansys HFSS Regions and Ansys SIwave geometry are integrated in a final circuit simulation that computes results for the entire design.

Note:

The Ansys SIwave Regions layer enables region-based simulations. Users may want to create a separate HFSS solution setup if they want to modify default HFSS settings that are used when solving HFSS regions. This HFSS simulation setup can then be associated with an Ansys SIwave region-based solution by selecting the HFSS setups/sweeps when configuring the Ansys SIwave sweep in the **Edit Frequency Sweep** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Methods of Defining HFSS Regions

There are two methods of defining HFSS regions. A standard method is used by default. An alternative method, using conformal radiation boundaries, can be selected by setting the environment variable SIWAVE_HFSS_REGION_RECESS_PORT_DISTANCE_FACTOR=<trace recess factor>. The trace recess factor can be any real value greater than 0, and defines how far, as a multiple of trace widths, the trace is recessed from the region boundary (e.g., setting SIWAVE_HFSS_REGION_RECESS_PORT_DISTANCE_FACTOR=2.5 will result in traces being recessed from the clipping extent by 2.5x the trace width).

The following table compares the two methods.

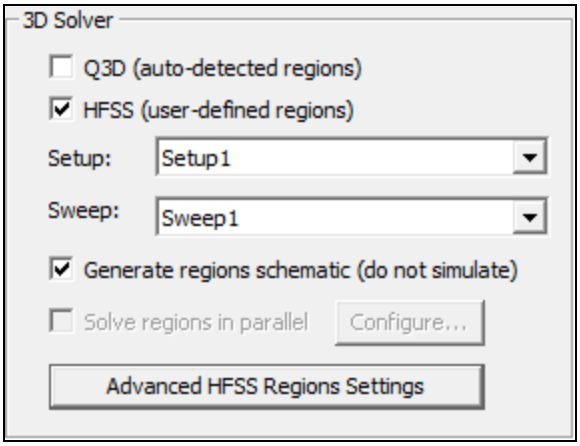
	Standard Method	**Conformal Radiation Boundaries**
Dielectric:	Forms a bounding box that extends 15% of width/length past the defined geometry.	Clipped to exactly match the board outline (a conformal clip with 0% padding).
Radiation Boundary / HFSS Airbox:	Extends 25% past defined geometry in a bounding box.	Clipped to exactly match the board outline.
Signal Traces:	Signal traces that require virtual ports are slightly rerouted to ensure they hit the boundary at an orthogonal angle.	Signal traces that require virtual ports are not clipped exactly at the region boundary; instead, their ends are moved slightly inside the boundary with no other rerouting. The distance is

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		defined by the trace recess factor.

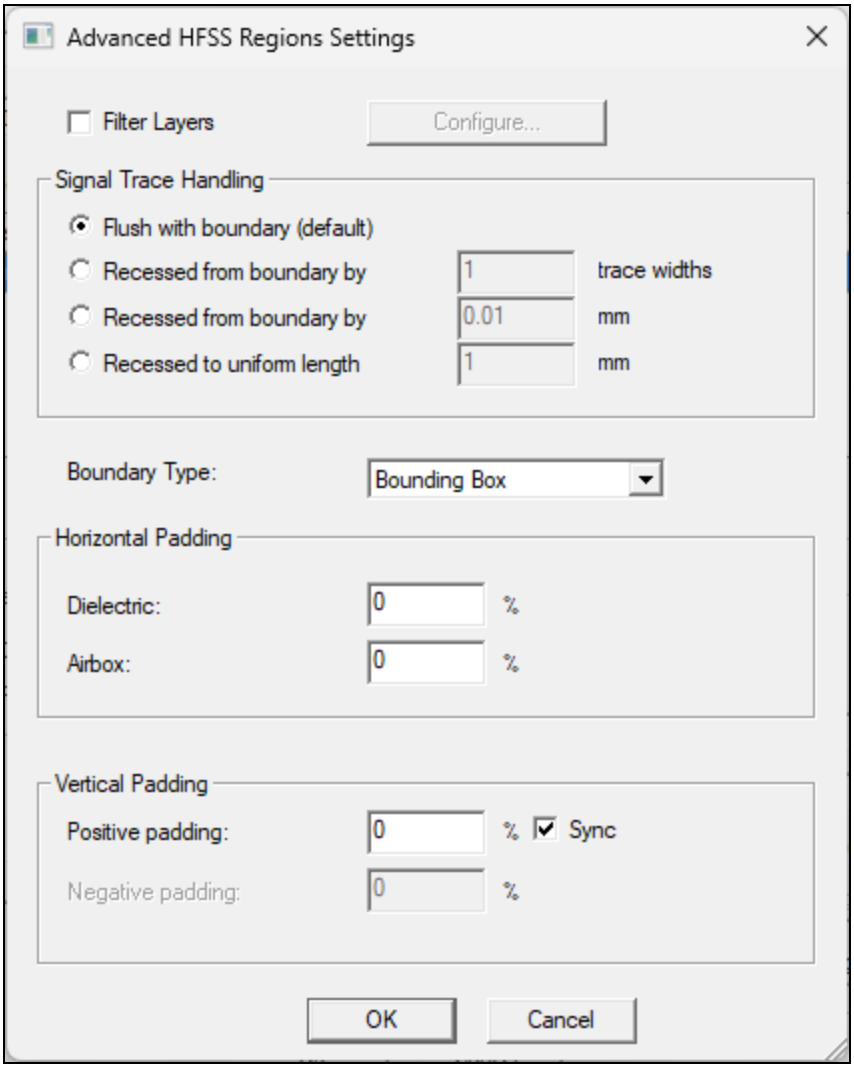
HFSS Regions for Ansys SIwave Sweeps

1. From the **Project Tree**, expand **Analysis**.
2. Expand **SIwave**, if one exists.
3. Double-click an Ansys SIwave sweep to open the **Edit Frequency Sweep** window.
4. From the **3D Solver** area, select **Q3D** and/or **HFSS**.



5. Using the **Setup** and **Sweep** drop-down menus, select an HFSS setup and a corresponding sweep.
6. If appropriate, uncheck the **Generate regions schematic (do not simulate)** box to activate **Solve regions in parallel** and **Configure**. Refer to Hierarchical Restrictions.
7. If appropriate, click **Advanced HFSS Regions Settings** to open the **Advanced HFSS Regions Settings** window. Use the **Advanced HFSS Regions Settings** window to configure simulation boundaries and trace behavior for user-defined HFSS regions in SIwave region-based simulations. These settings affect how dielectric and airbox extents are clipped, how trace ends are recessed, and how vertical padding is applied. Refer to Ansys SIwave™ Region-Based Simulations in HFSS 3D Layout.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. To enable layer filtering, check the **Filter Layers** box. Then click **Configure** to open a **Layer Selection** window. Manually choose which signal layers are included in the HFSS region definition by checking adjacent boxes. Only selected layers will be included when defining HFSS regions. Click **OK** to confirm selection and close the **Layer Selection** 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:**
>
> Filtering layers helps exclude irrelevant geometry and ensures that only signal nets intended for simulation are included in the HFSS region. Unselected layers will be excluded from simulation, and all geometry on those layers will be deleted. If all layers containing relevant signal geometry are not included, path discontinuities may result. Path discontinuity may also result if relevant GND layers are excluded (typically those immediately above and below signal layers). For example, if signal geometry exists on the Top, Bottom, and Layer 7, you should also include Layer 1 (below Top), Layer 2 (above Bottom), and Layers 6 and 8 (adjacent to Layer 7). The full set would be: Layer 1, Layer 2, Layer 6, Layer 7, Layer 8, Layer N-1, Layer N, et cetera.

9. From the **Signal Trace Handling** area, choose how signal traces are recessed from the region boundary:
    - **Flush with boundary (default)**: Virtual ports are placed directly on the region boundary.
    - **Recessed from boundary by**: Traces are clipped to create a gap between the virtual ports and the boundary. The recess is specified as a multiple of trace widths.
    - **Recessed from boundary by**: Traces are clipped to create a gap between the virtual ports and the boundary. The recess is specified in millimeters.
    - **Recessed to uniform length**: Traces are clipped so the remaining trace length is uniform across all traces in millimeters.

> **Note:**
>
> Virtual ports must lie entirely within the region to be simulated. In the case of virtual ports generated flush to the boundary, trace routes are modified and a generous airbox is required to meet this constraint. For simulations with tight, conformal dielectric and airboxes, virtual ports must be recessed from the boundary to meet this constraint.

10. From the **Boundary Type** drop-down menu, select the clipping method for both dielectric and airbox boundaries:
    - **Bounding Box**: Adds horizontal and vertical padding beyond the geometry.
    - **Conformal**: Clips exactly to the board outline (i.e., 0% padding).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. From the **Horizontal Padding** area, enter values for:
    - **Dielectric**: Percentage of horizontal padding added around the dielectric region.
    - **Airbox**: Percentage of horizontal padding added around the airbox.

    > **Note:**
    >
    > If **Flushed with boundary (default)** is selected, the boundary type is forcibly set to **Bounding Box**. A generous horizontal and vertical padding of approximately **15–25%** is recommended. If **Recessed from boundary by** or **Recessed to uniform length** is selected, the boundary type is forcibly set to **Conformal**.

12. From the **Vertical Padding** area:
    - Enter values for **Positive padding** and **Negative padding**, as percentages.
    - Check the **Sync** box to apply the same value to both directions.

    > **Note:**
    >
    > Vertical padding defines the extent of the simulation region above and below the board. Excessive padding may increase simulation time.

13. Click **OK** to apply the settings or **Cancel** to discard changes.

Rational Interpolation

Between the actual solution frequencies, the frequency response is obtained by rational interpolation. Ansys SIwave adaptively selects the frequency points at which it computes the field solution. After a new frequency point is solved, a new interpolating fit is generated. This is compared to the interpolant on the previous step, and the maximum difference between the two is determined. If the difference exceeds the requested tolerance, then a new frequency point is selected for a solution. The interpolating sweep is complete when the difference between successive interpolants is less than the error tolerance criterion.

1. From the **Frequency Sweep** grid, select the method for distributing points on the **Distribution** option. Select one of the following:
    - **Linear Count** – The difference between the start frequency and the stop frequency is calculated and is divided by the number of solution points.
    - **Linear Step** – The start frequency is incremented using the step size until the stop frequency to calculate the number of points.
2. Enter the value for the minimum frequency to sweep in the **Start** field.
3. Enter the value for a sweep maximum frequency in the **Stop** field.

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 **Points**, **Stepsize** or **Samples**, depending on the **Distribution** option selected.
5. Optionally, modify the frequency points: click **Add Above**, **Add Below**, or **Delete Selection**.
6. To preview all the frequency points, click **Preview**. The **Frequency List Preview** window opens, showing all the points.
7. Select **Time Domain Calculation** to export the full-wave SPICE subcircuit. The **Start** value changes to **0 Hz**.

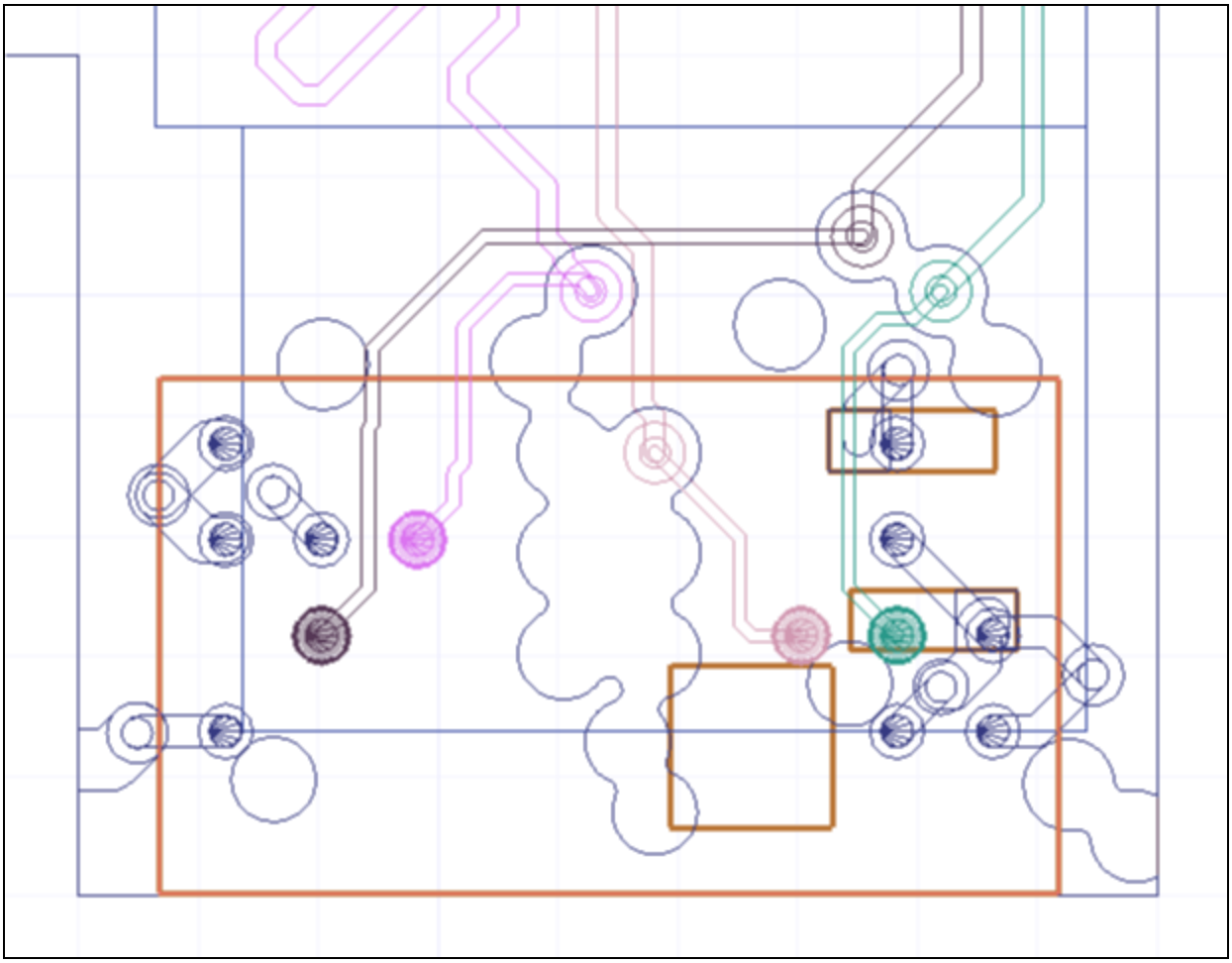
> **Note:**
>
> If there is an existing S-parameter solution, an FWS simulation can be run without needing a 0Hz or 1Hz point.

Hierarchical Restrictions

If the project contains a hierarchical layout, ensure the following:

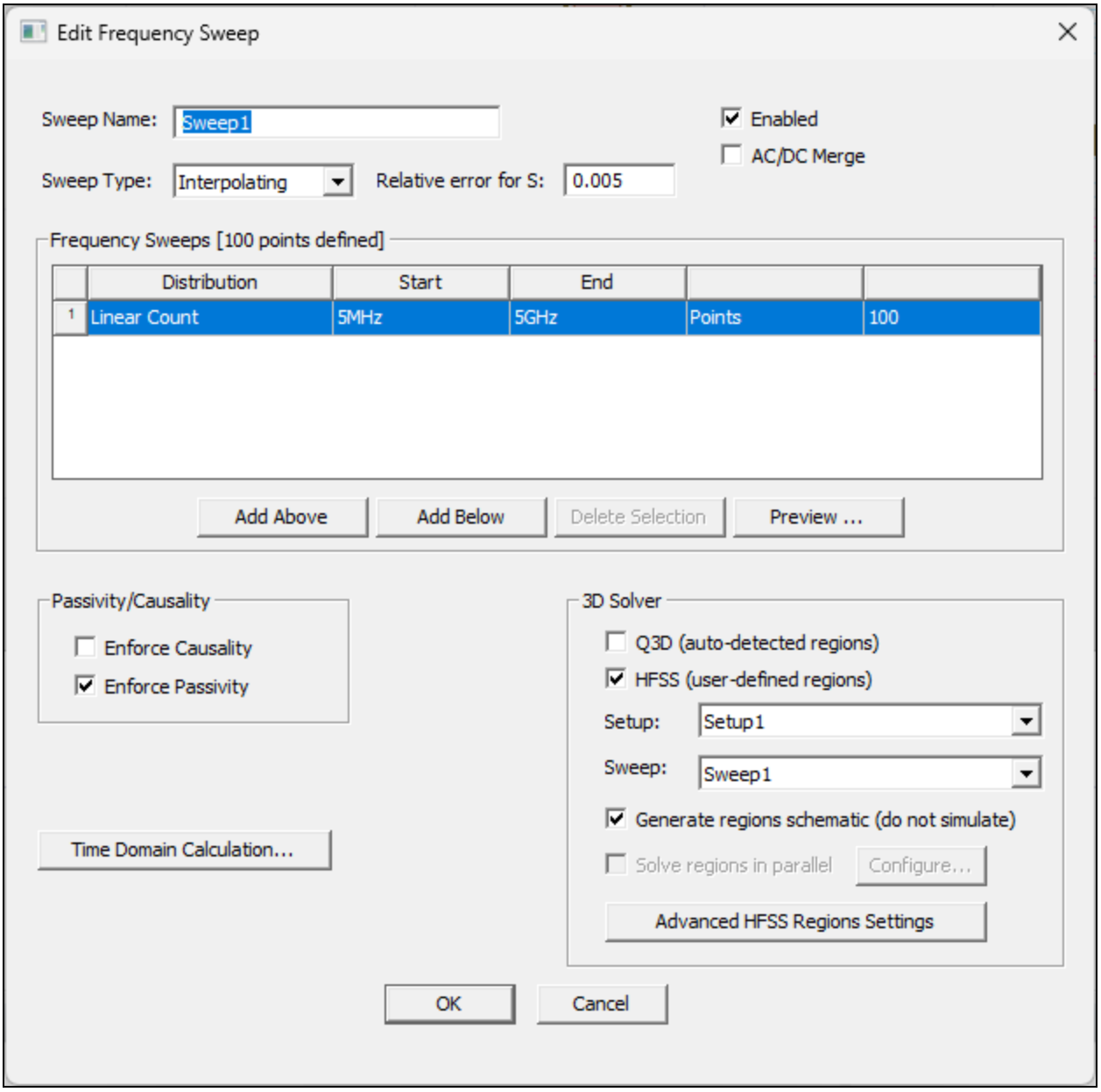
- HFSS regions must surround the points of contact between both cells. Refer to Computing SYZ Solutions in HFSS 3D Layout.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The **HFSS (user-defined regions)** box is checked. If applicable, check either the **Generate regions schematic (do not simulate)** box or the **Solve regions in parallel** box. Refer to Generating a Regions Schematic in HFSS 3D Layout and Configuring a Parallel HFSS Regions Simulation in HFSS 3D Layout.

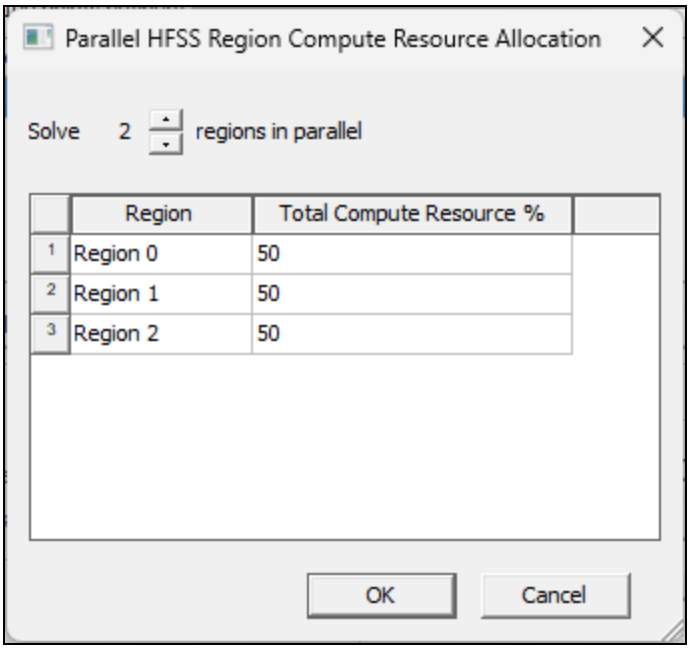
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- If **Solve regions in parallel** is checked, click **Configure** to open the **Parallel HFSS Region Compute Resource Allocation** window. Specify the number of regions to solve in parallel (e.g., **2**) and allocate a percentage of total resources to each region (e.g., **50**). The percentage values are converted to specific machine names, CPU counts, and memory footprints during simulation. Then click **OK** to close the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel HFSS Region Compute Resource Allocation window.



Generating a Regions Schematic in HFSS 3D Layout

From the SIwave sweep setup, you can generate an HFSS regions schematic. This affords greater control and visibility into the HFSS region setups.

Each HFSS region is modeled as a separate 3D Layout design under a single 3D Layout project. There is an additional design showing what is left after the other regions are removed. Once these 3D Layout projects have been instantiated in AEDT, a schematic design is generated. A Nexxim LNA sweep setup (that matches the S-parameter sweep) is inserted and the schematic is ready for simulation. Prior to solving, you can examine and modify each associated 3D Layout project.

> **Important:**
>
> Modifications to ports at region interfaces, if done carelessly, can invalidate the schematic level connections and requires user intervention to rectify.

To generate an HFSS regions schematic:

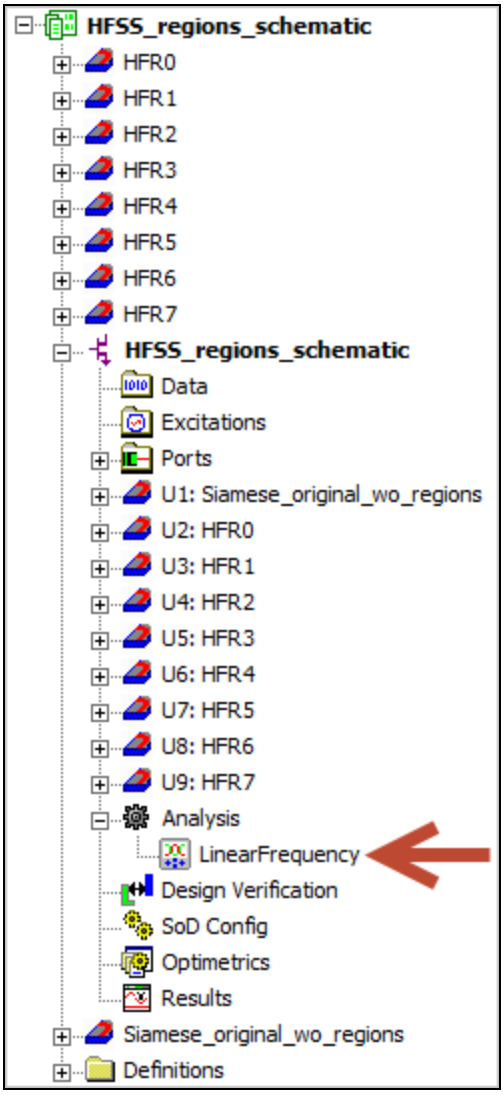
1. Set up an Ansys SIwave™ regions simulation.
2. From the Edit Frequency Sweep window, click **HFSS (user-defined regions)** and **Generate regions schematic (do not simulate)**.

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 **Generate regions schematic** option is unavailable, ensure that **Solve regions in parallel** is deselected. These options are mutually exclusive.

3. Click **OK**.
4. Right-click the simulation and select **Analyze**.

Instead of running a simulation, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform generates the schematic. It looks similar to the following:



Note the location of the LNA sweep setup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

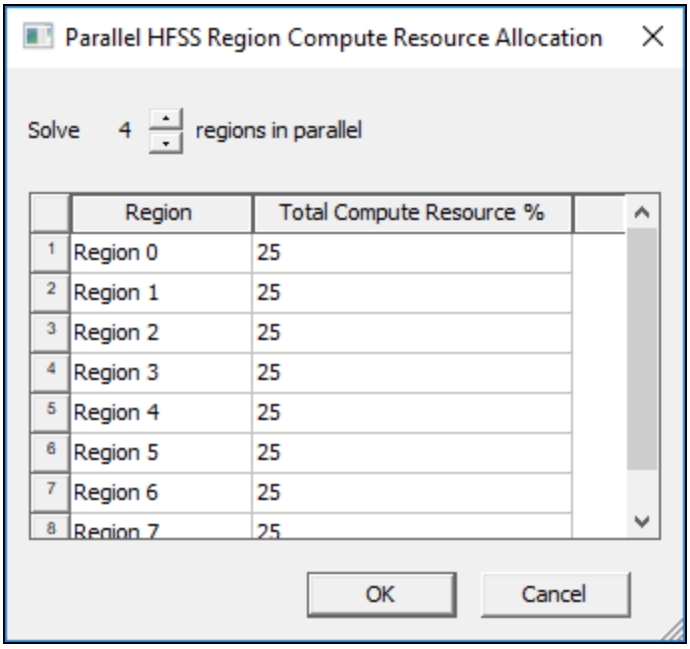
Configuring a Parallel HFSS Regions Simulation in HFSS 3D Layout

From the Ansys SIwave™ sweep setup, follow these steps to configure a parallel solve.

1. Configure a distributed solution setup.
2. Set up an Ansys SIwave™ regions simulation.
3. From the **Edit Frequency Sweep** window, click **HFSS (user-defined regions)** and **Solve regions in parallel**.

   > **Note:**
   >
   > If the **Solve regions in parallel** option is unavailable, ensure that **Generate regions schematic** is deselected. These options are mutually exclusive.

4. Click **Configure** to open the **Parallel HFSS Region Compute Resource Allocation** window.



5. Specify the number of regions to solve in parallel, and allocate a percentage of total resources toward each region. See the following example.

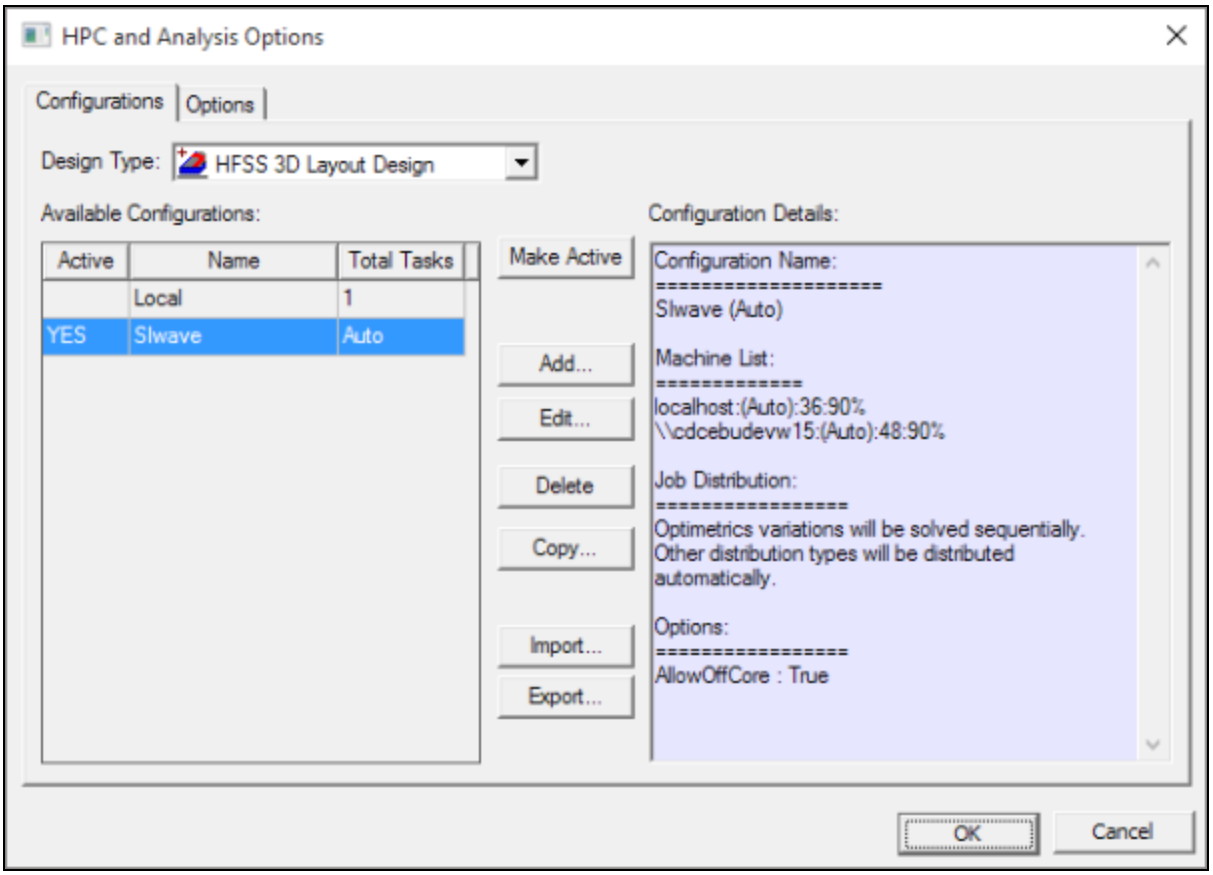
   The percentage values are converted to specific machine names, CPU counts and memory footprints during simulation setup.

6. Click **OK** to return to the **Edit Frequency Sweep** window.
7. Click **OK** to close 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.

Example

Suppose **localhost** has been configured to use 36 cores and 90% of available memory, and **cdcebudevw15** has been configured to use 48 cores and 90% of available memory:



In total, there are 84 cores at our disposal, so each region configured to use 25% of total available resources gets 21 cores.

Memory is divided among the regions using the following formula:

HFSS region memory % = (% memory to use on machine) X (#cores to use for region)/(#cores available on machine)

Cores are allotted to regions starting with the first machine in the distributed solve setup (localhost, in this example). Once all cores on that machine have been exhausted, cores are pulled on the next machine in the list (cdcebudevw15, in this example). Once all cores on all machines have been exhausted, the solver loops back to the first machine and starts again.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To avoid resource contention, you must balance compute resource percentage assignments for each region with the number of regions being solved in parallel. Working through this core/memory assignment logic,ending up with the following:

- **Region 0** – are allotted 21 cores from localhost; % memory = 90% X (21/36) =52.5%

  This leaves 15 remaining cores on localhost and 48 on cdcebudevw15.

- **Region 1** – are allotted 15 cores from localhost and 6 cores from cdcebudevw15 (21 total cores)

  This leaves 0 remaining cores on localhost and 42 remaining cores on cdcebudevw15.

  % memory on localhost = 90% * (15/36) = 37.5%

  % memory on cdcebudevw15 = 90% * (6/48) = 11.25%

- **Region 2** – are allotted 21 cores from cdcebudevw15; % memory = 90% X (21/48) = 39.375%

  This leaves 21 remaining cores on cdcebudevw15.

- **Region 3** – are allotted 21 cores from cdcebudevw15; % memory = 90% X (21/48) = 39.375%

  All cores on all machines have been exhausted at this point, so the solver goes back to the first machine in the distributed solve setup list...

- **Region 4** – are allotted 21 cores from localhost; % memory = 90% X (21/36) =52.5%

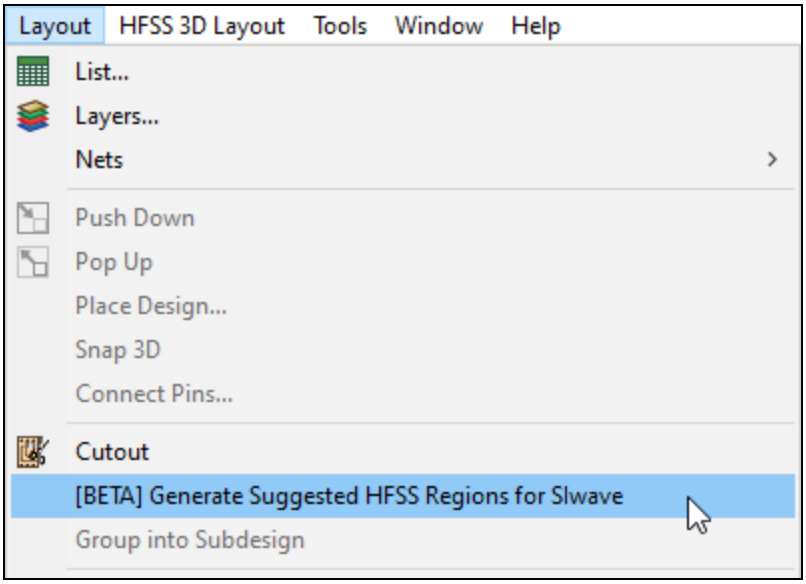
Generating Automatic HFSS Regions for Ansys SIwave™ (Beta)

HFSS 3D Layout can automatically add Ansys HFSS™ regions for Ansys SIwave™ simulation. These regions are added in locations of significant 3D activity, currently limited to geometry surrounding user-defined excitations and vias on signal nets along routes connecting two user-defined excitations. The Auto Regions algorithm iteratively expands these initial regions to ensure that the ports generated automatically at the SIwave/HFSS boundaries during simulation are electrically coherent (i.e., there are no discontinuities in their current return paths). For smaller designs, this algorithm should run instantaneously. For larger PCBs, it may take longer.

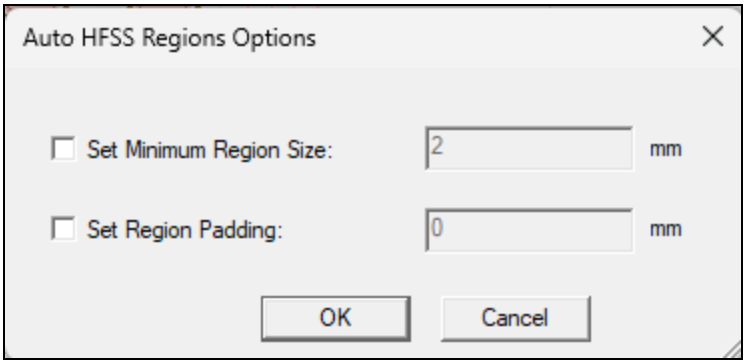
Complete the following steps to automatically generate HFSS regions.

1. From **Layout**, select **Generate Suggested HFSS Regions for SIwave** to open the **Auto HFSS Regions 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.



2. From the **Auto HFSS Regions Options** window, do the following:

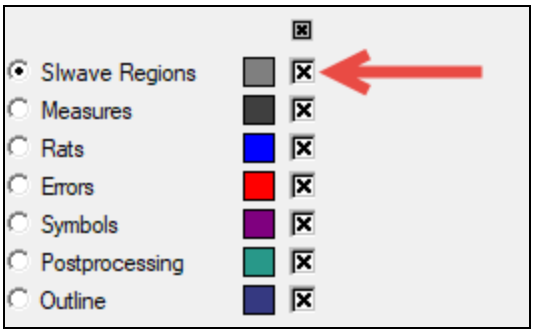


- If appropriate, check **Set Minimum Region Size** to enforce a minimum size for each automatically generated HFSS region. Enter a value in millimeters (default is **2 mm**).
- If appropriate, check **Set Region Padding** to expand the region boundary around critical GND vias. Enter a padding value in millimeters (default is **0 mm**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Auto Regions identifies key GND vias that must be included in the HFSS region. By default, the boundary snaps tightly around these elements. The **Region Padding** option expands the boundary outward to include additional clearance around them, which may improve simulation robustness.

3. View the generated regions on the **SIwave Regions** layer.



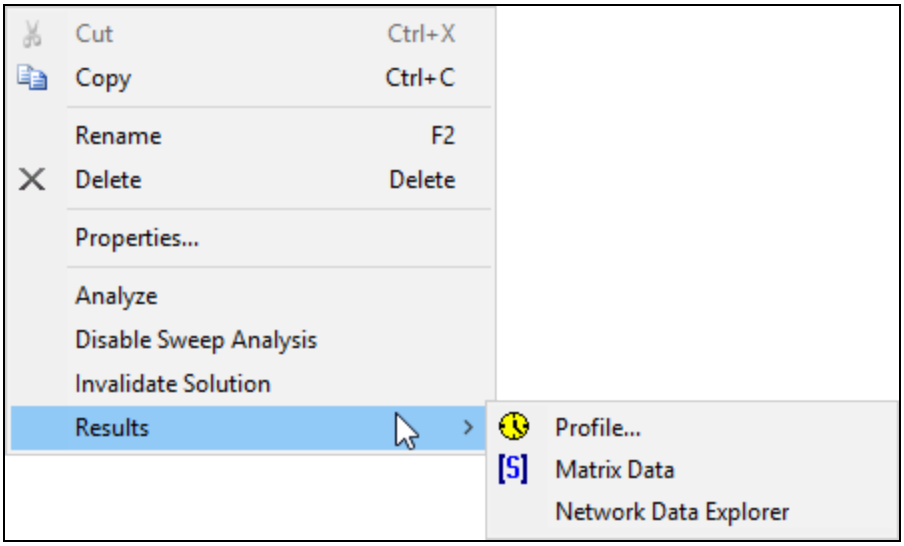
Viewing and Exporting SYZ Solution Results

To view SYZ results:

1. Under **Analysis** in the **Project Manager** window, expand the Ansys SIwave™ 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.

2. Right-click your selected **Sweep** and select **Results**.



3. Select the result type you want to view.

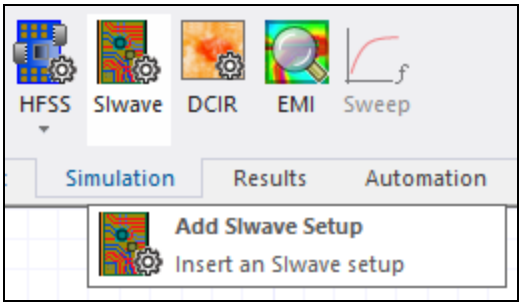
   Results menu options include:

   - **Profile** – opens the simulation profile.
   - **Matrix Data** – opens the Matrix Data, which you can export. Also export equivalent circuit data.
   - **ndExplorer** – opens the simulation results in ndExplorer.

Configuring S-Parameters in an Ansys SIwave™ Solution Setup (Example)

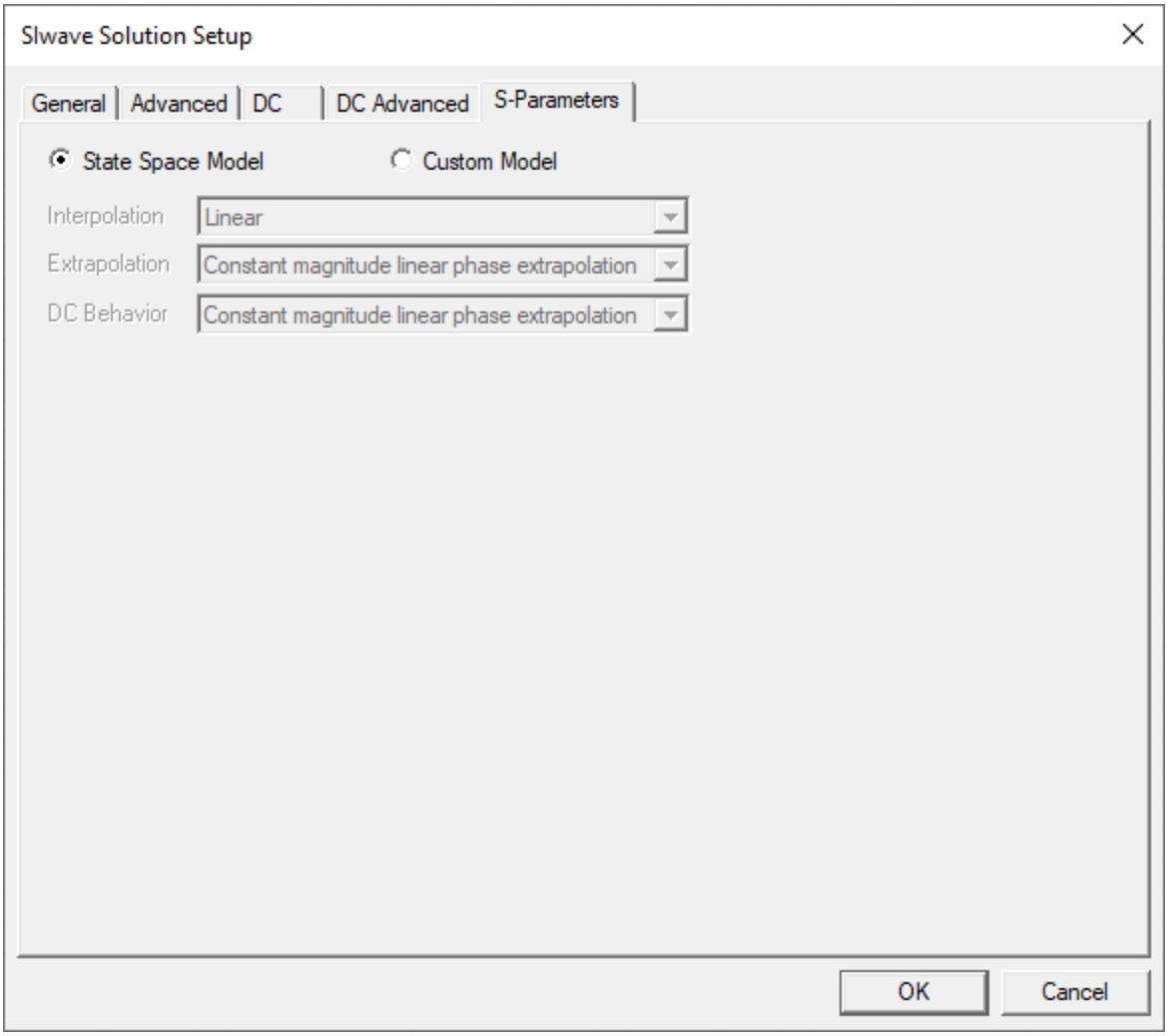
Follow these steps to configure S-parameters from the **SIwave Solution Setup** window.

1. From the **Simulation** tab, click **SIwave** to open the **SIwave Solution Setup** window.



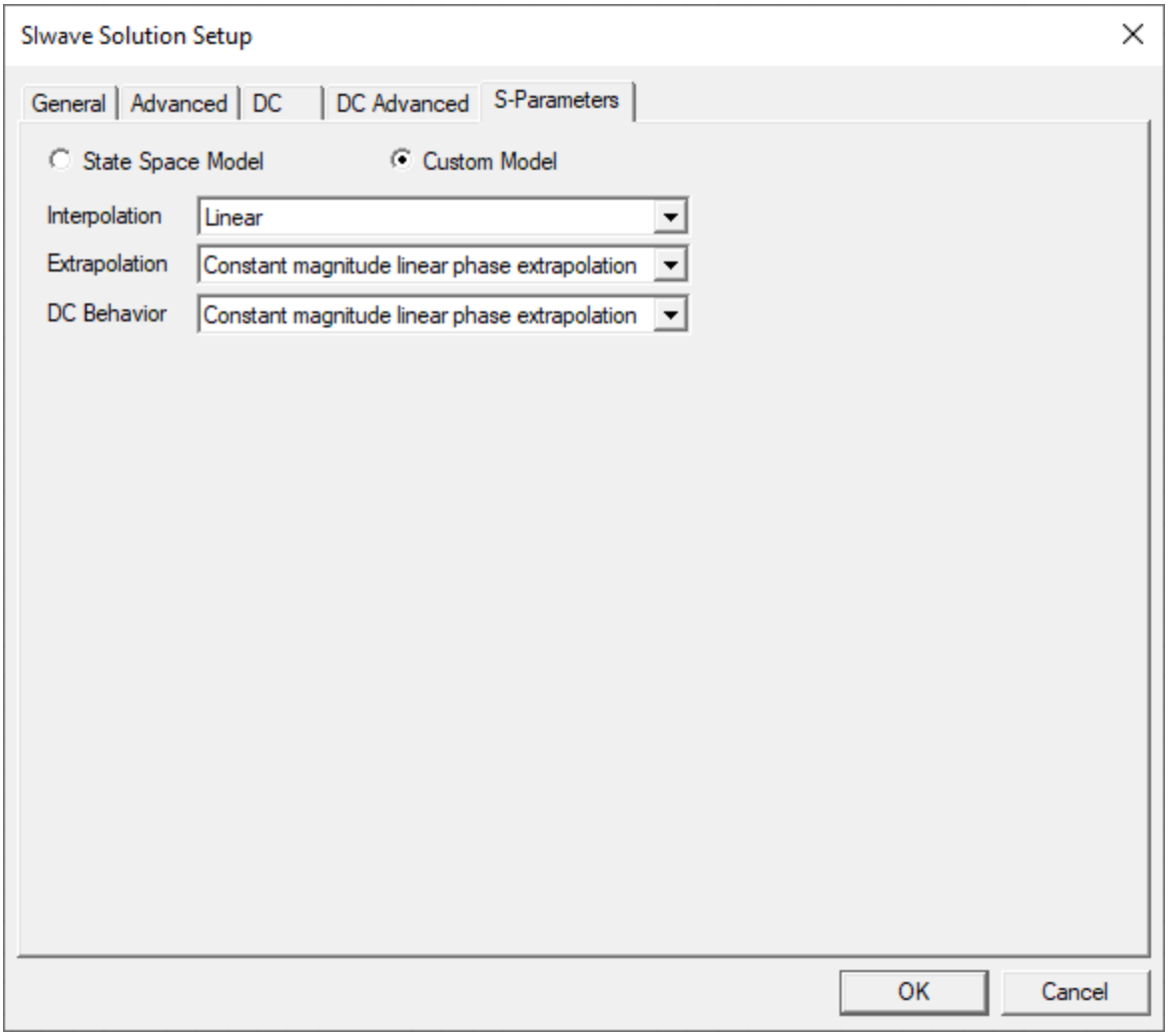
2. Navigate to the **S-Parameters** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



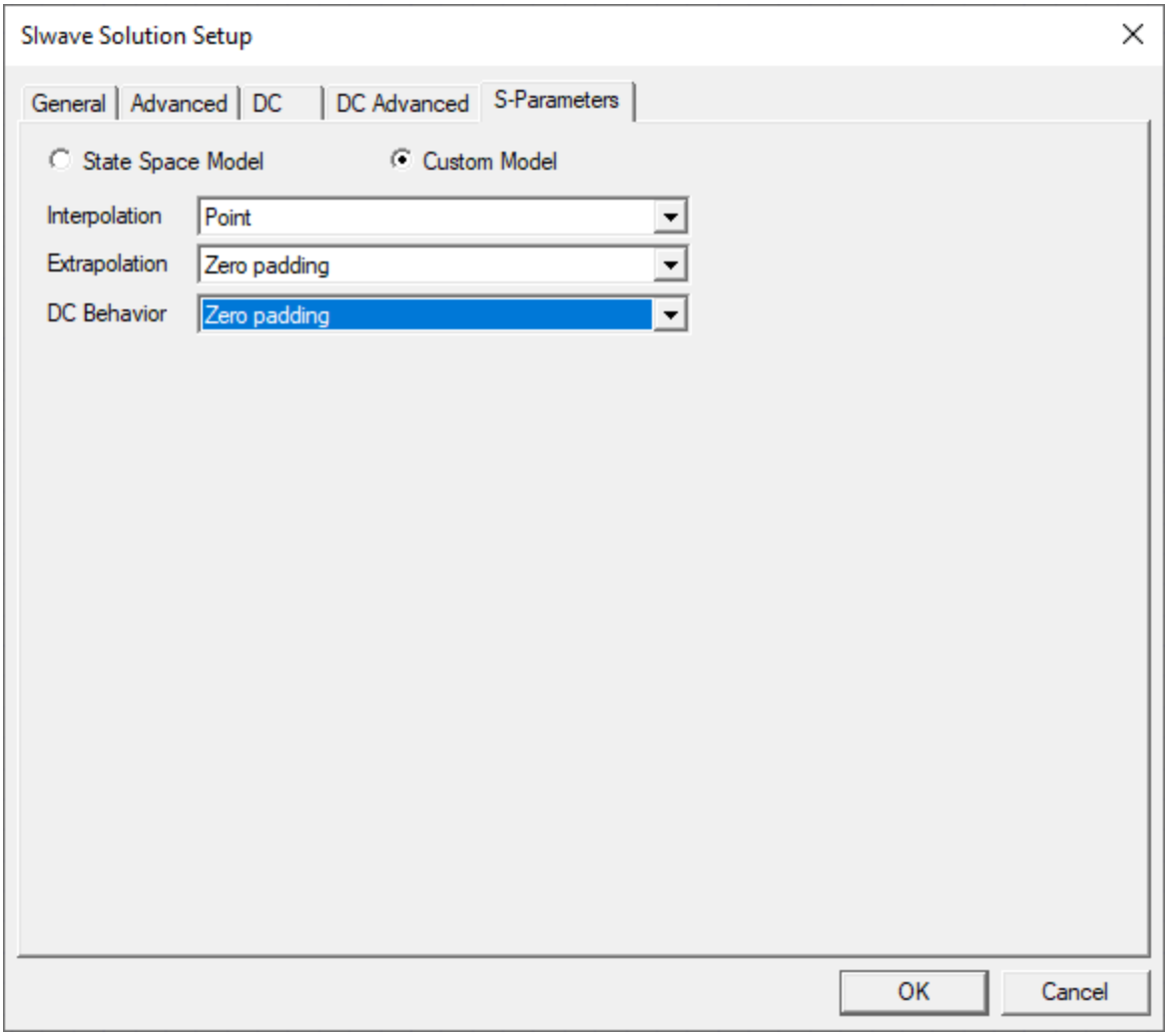
3. Select **Custom Model**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



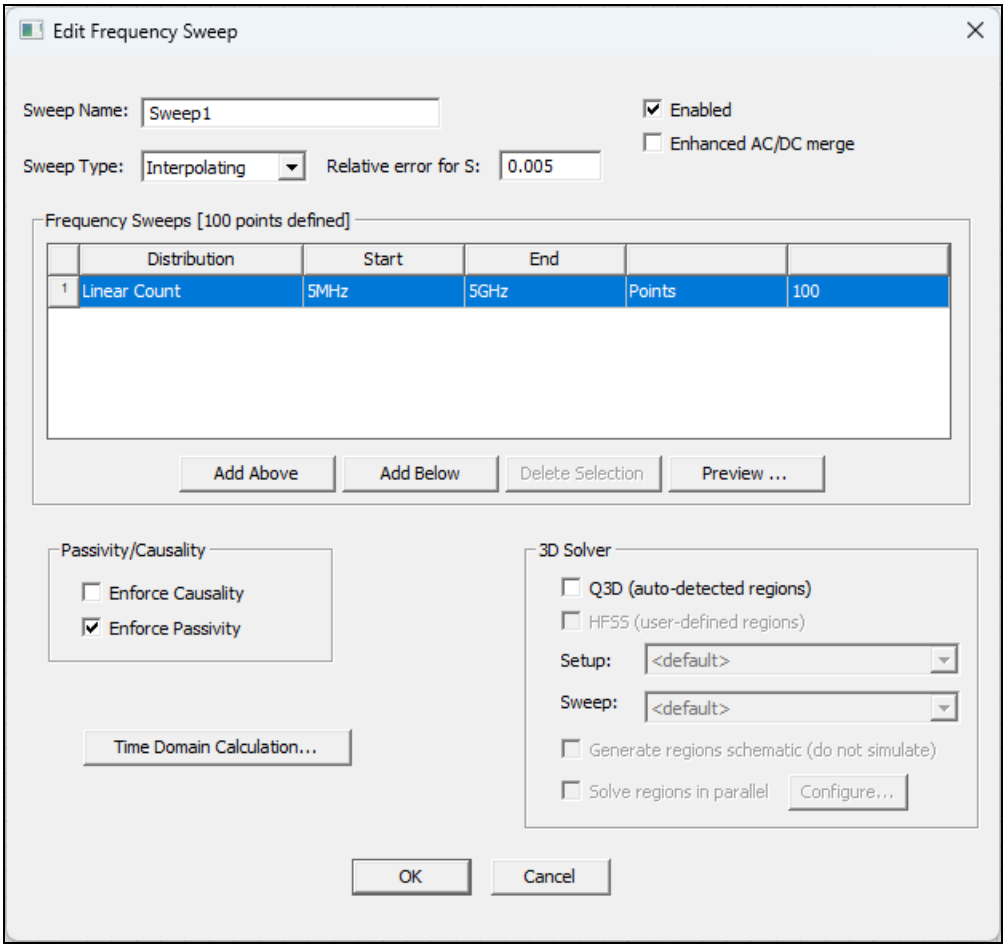
4. Set options from the following drop-down menus:
    - **Interpolation** – select an interpolation method for data points in the frequency range of the Touchstone data.
    - **Extrapolation** – select the extrapolation method for data points outside the frequency range of the Touchstone data.
    - **DC Behavior** – select the behavior of the N-Port at DC (zero frequency).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



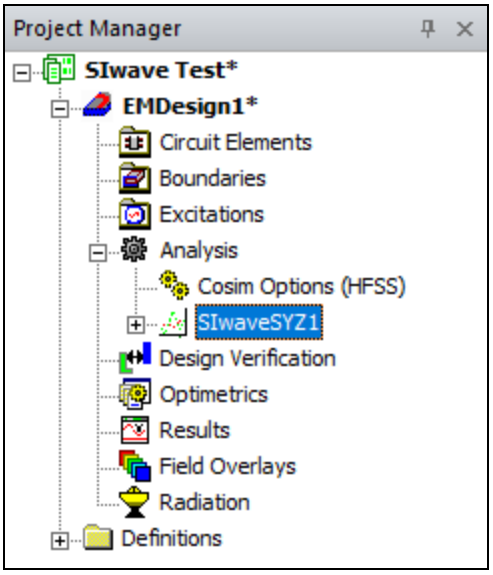
5. Click **OK** to save changes and close the **Slwave Solution Setup** window. The **Edit Frequency Sweep** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Configure the frequency sweep, as required.
7. To make further changes to the parameter options, return to the same setup by navigating to the **Project Manager** window. Then expand the **Project Tree** > **[*active design folder*]** > **Analysis** and double-click the Ansys SIwave™ setup (e.g., **SIwaveSYZ1**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Computing Near Field Solutions in HFSS 3D Layout (General Mode Only)

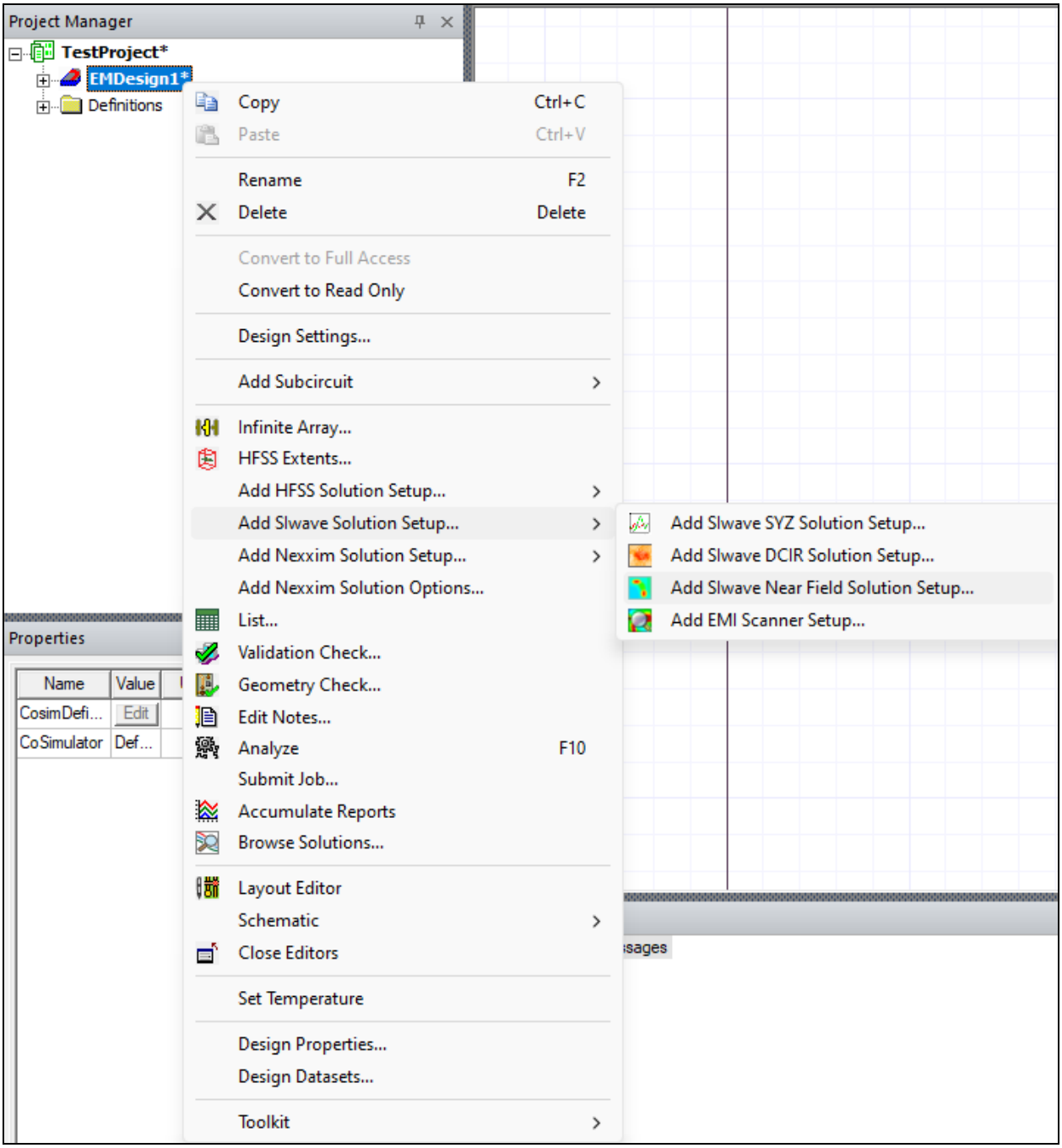
To analyze the radiated fields associated with a design, first define a radiation surface. The values of the fields calculated over this surface are used to compute the fields in the space surrounding the device. This space is typically split in two regions: near field and far field. The near field region exists at less than a wavelength from an energy source.

Setting Up a Near Field Simulation

Follow these steps to add a near field solution setup to an active design.

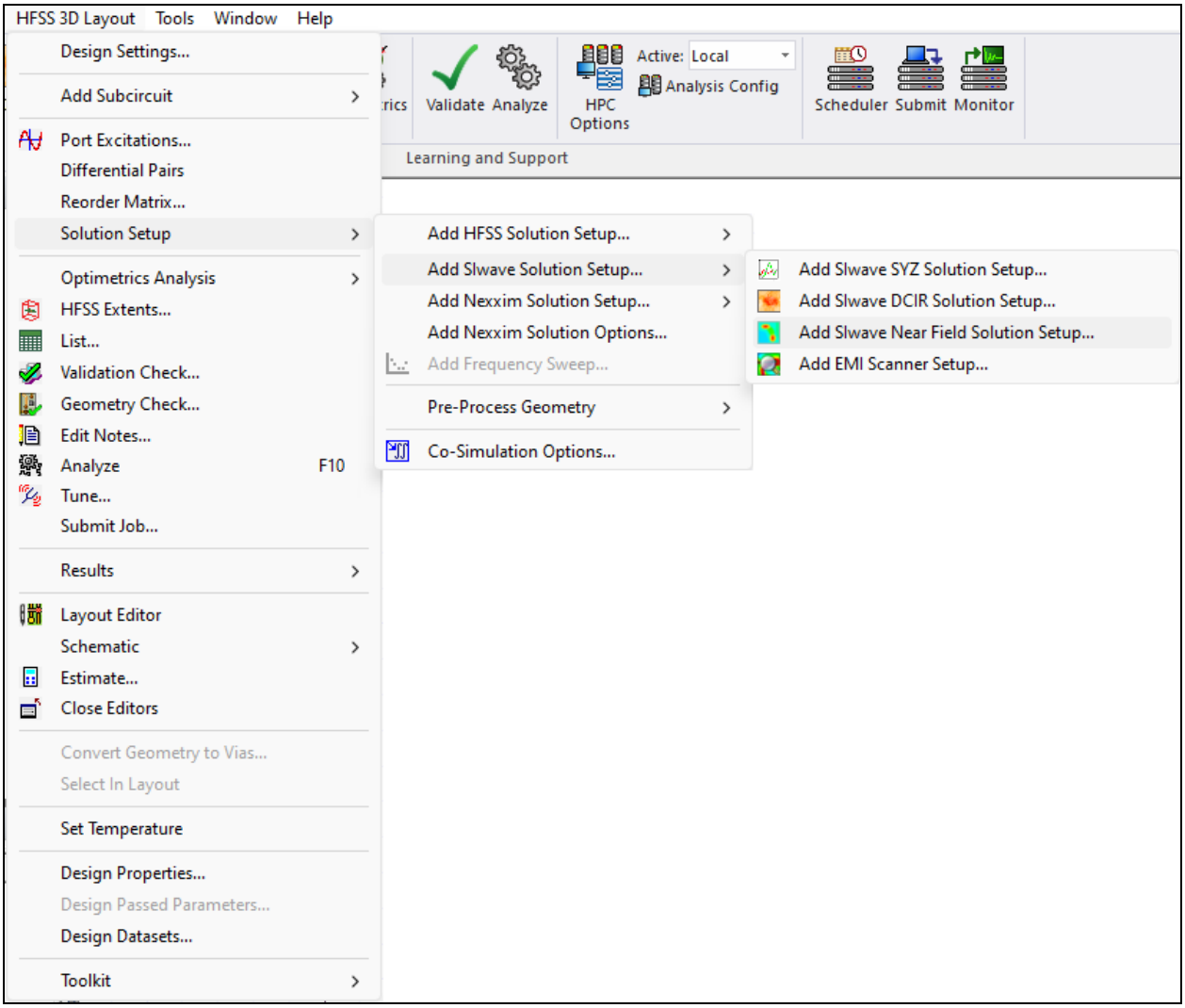
1. Open the **Near Field Solution Setup** window by doing one of the following:
   - From the **Project Manager** window, expand the **Project Tree** (e.g., **TestProject**). Then right-click the active design folder (e.g., **EMDesign1**) and select **Add SIwave Solution Setup** > **Add SIwave Near Field 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.



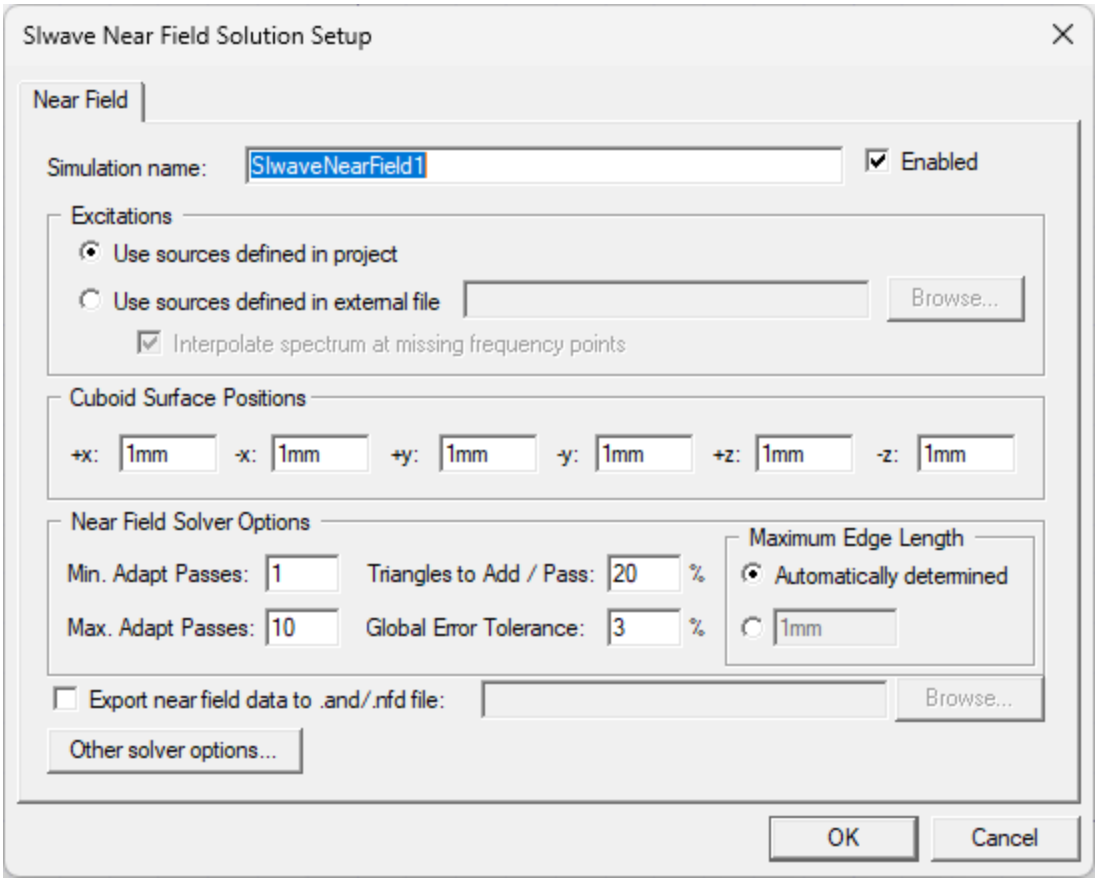
- From **HFSS 3D Layout**, select **Solution Setup** > **Add Slwave Solution Setup** > **Add Slwave Near Field Solution Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Near Field Solution Setup** window, make selections to change the following default values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

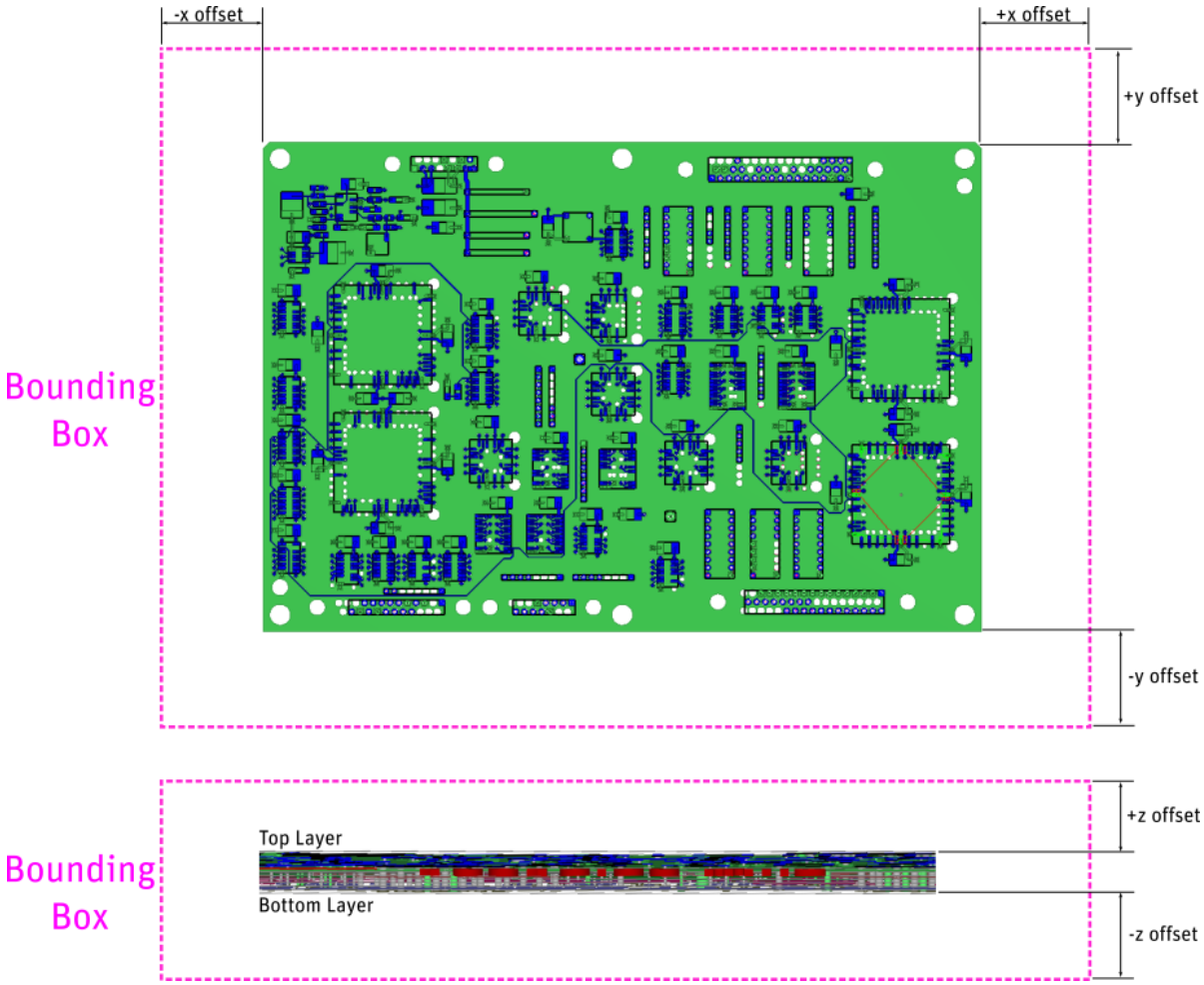


3. **Simulation name** — enter a name in the field or accept the default (i.e., **SIwaveNearField1**).
4. From the **Excitations** area, select a source of excitations (i.e., **Use sources defined in the project** or **Use sources defined in external file**). If users select **Use source defined in external file**, **Browse** to an appropriate file and select it (e.g., **test.tmp**). If applicable, uncheck the **Interpolate spectrum at missing frequency points** box.

> **Note:**
>
> A transient excitation source from either Circuit or HFSS 3D Layout sources can be assigned after setup is complete. Refer to Assigning a Transient Excitation.

5. From the **Cuboid Surface Positions** area, enter values for **+x**, **-x**, **+y**, **-y**, **+z**, and **-z** offset (e.g., **5mm**). These values define a cuboid that completely encloses the design by specifying the cuboid face offset values from the design bounding box. The near field is evaluated on the surface of this cuboid, as shown in 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.



> **Note:**
>
> Offsets that are very small may increase simulation time.

6. From the **Near Field Solver Options** area, enter values for the following:
   a. **Min. Adapt Passes** — an adaptive analysis will not stop unless the minimum number of passes specified is completed (e.g., **5**), even if convergence criteria are met.
   b. **Max. Adapt Passes** — the maximum number of mesh refinement cycles Ansys SIwave™ printed circuit board (PCB) and package electromagnetics simulation software will perform. Default is **10**. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops; otherwise, the adaptive analysis will continue unless the convergence criteria are reached.

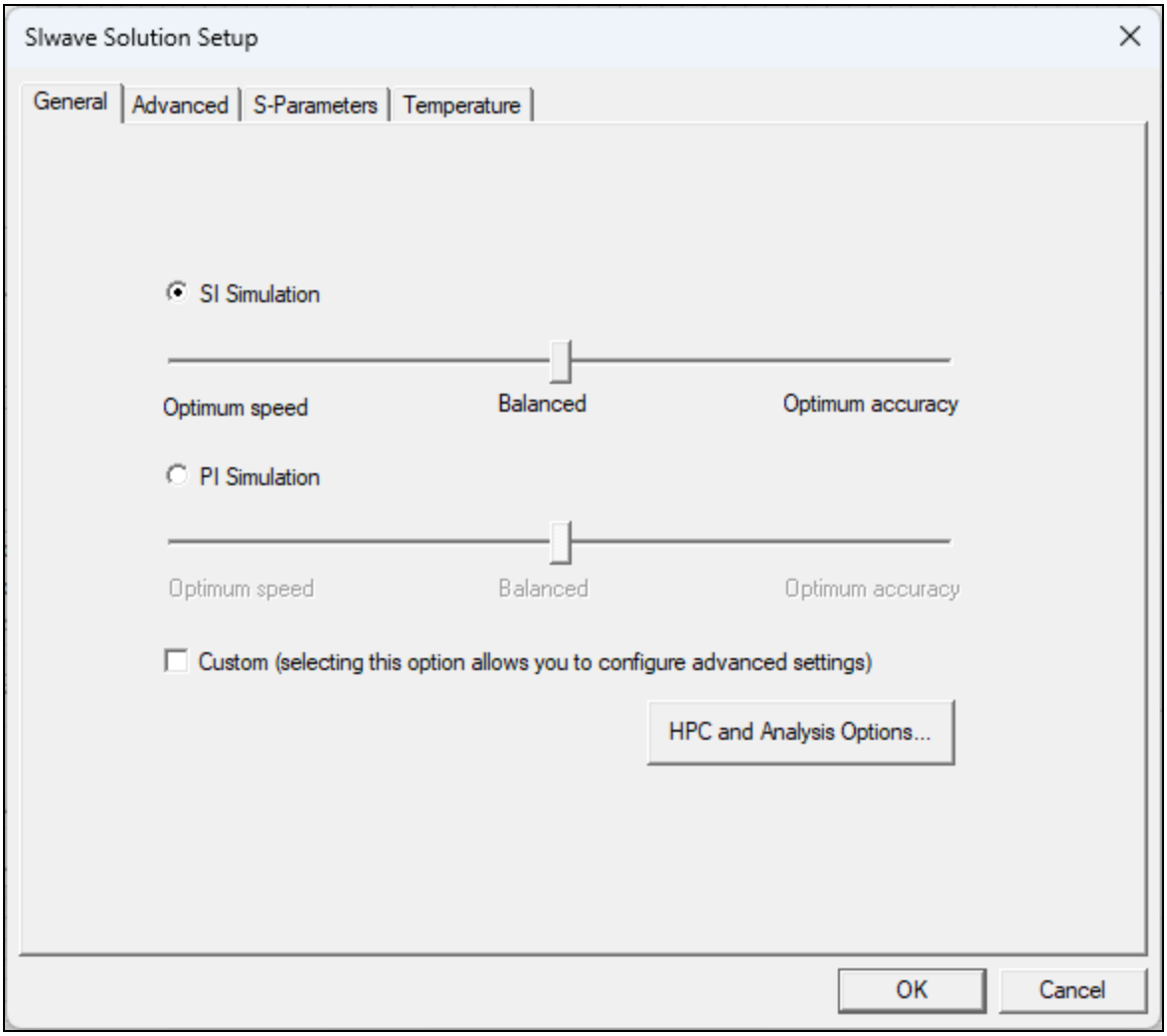
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The size of the finite element mesh and the amount of memory required to generate a solution increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in Ansys SIwave requesting more memory than is available or taking excessive time to compute the solutions.

   c. **Triangles to Add/Pass** — default is **20**.
   d. **Global Error Tolerance** — maximum relative difference allowed between two successive passes. Default is **3**.

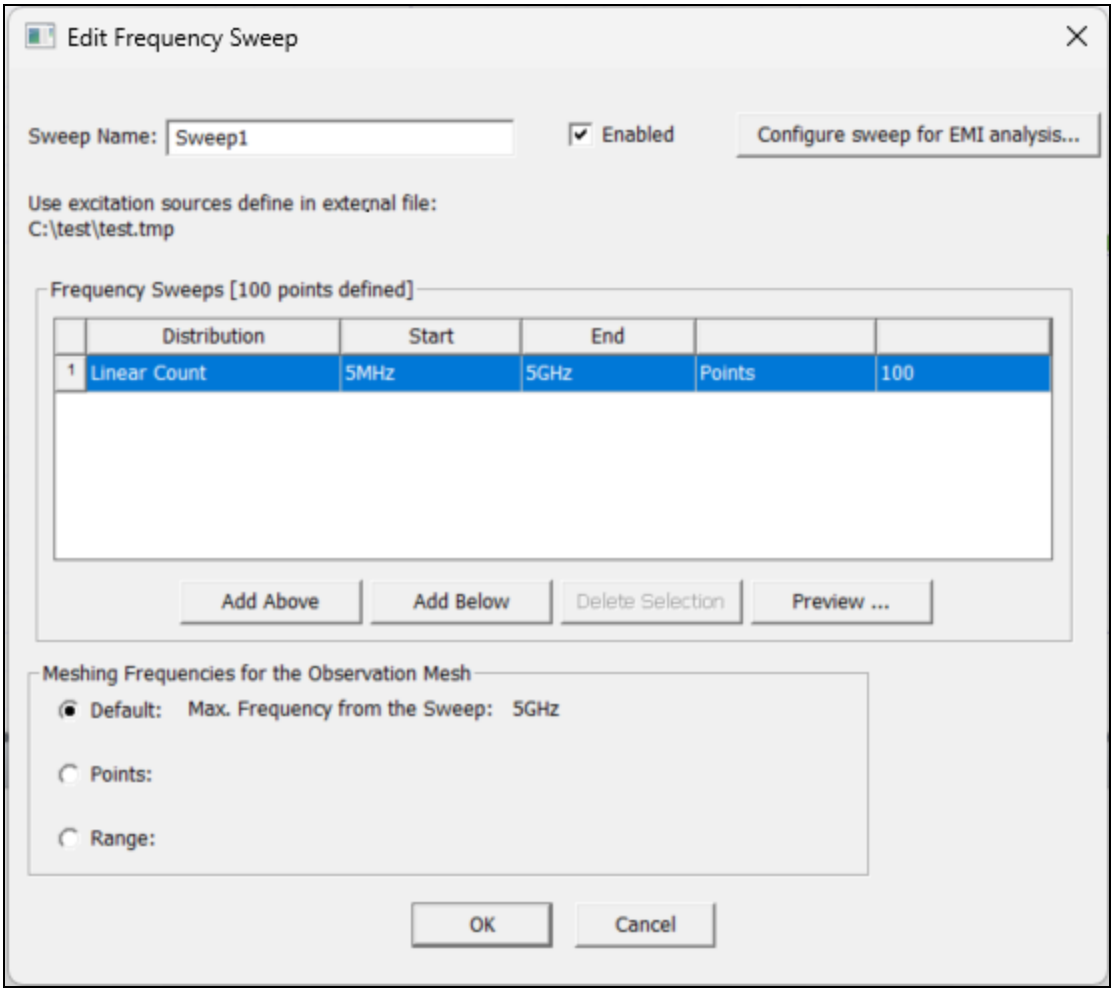
7. From the **Maximum Edge Length** area, select **Automatically determined** or enter a value in mils (e.g., **5mm**). Ansys SIwave generates a surface triangular mesh such that the triangle edge lengths are equal to or less than the specified value.
8. If applicable, check the **Export near field data to .and/.nfd file** box, then click **Browse**. Navigate to an appropriate directory to **Save** the file (e.g., **C:\ansysdev\test**). Near field data is exported in Ansys Nearfield Data (i.e., *.and) format, and can be used as external incident field data in Ansys HFSS simulations.
9. If applicable, click **Other solver options** to open the SIwave Solution Setup window. Otherwise, click **OK** to add the new near field solution setup 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.



The **Edit Frequency 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.



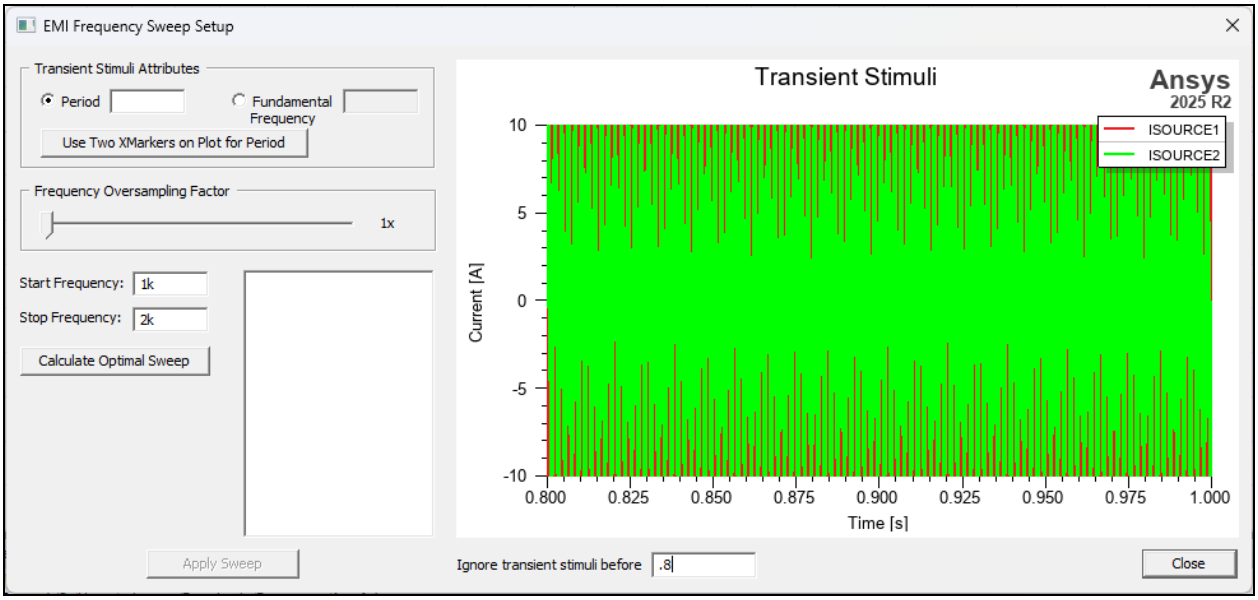
Setting Up a Frequency Sweep

Follow these steps to add a frequency sweep to a near field solution setup.

1. From the **Edit Frequency Setup** window, make selections from the following default values:
    - **Sweep Name** — enter a name in the field or accept the default (i.e., **Sweep1**).
    - **Enabled** — checked by default to enable the sweep.
    - **Configure sweep for EMI analysis** — click to access the **EMI Frequency Sweep Setup** 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:**
>
> EMI analysis is only available if the user selects **Use sources defined in external file** in the **Excitations** area of the **SIwave Near Field Solution Setup** window.



- From the **EMI Frequency Sweep Setup** window, make selections from the following default values:
  - From the **Transient Stimuli Attributes** area, select **Period** or **Fundamental Frequency** and enter an appropriate value in the field (e.g., **0.01**). The period is the measured length of information (T). The Fundamental Frequency is defined as $f_0 = 1/T$ where T is the period information.

    > **Note:**
    >
    > To use the **Use Two XMarkers on Plot for Period** function to automatically determine the period, refer to .

  - **Frequency Oversampling Factor** – move the slider to select the frequency range sampling density (e.g., **1x**). A higher oversampling factor results in more frequency points being introduced between the selected sweep start and stop frequencies, and may result in longer simulation time.

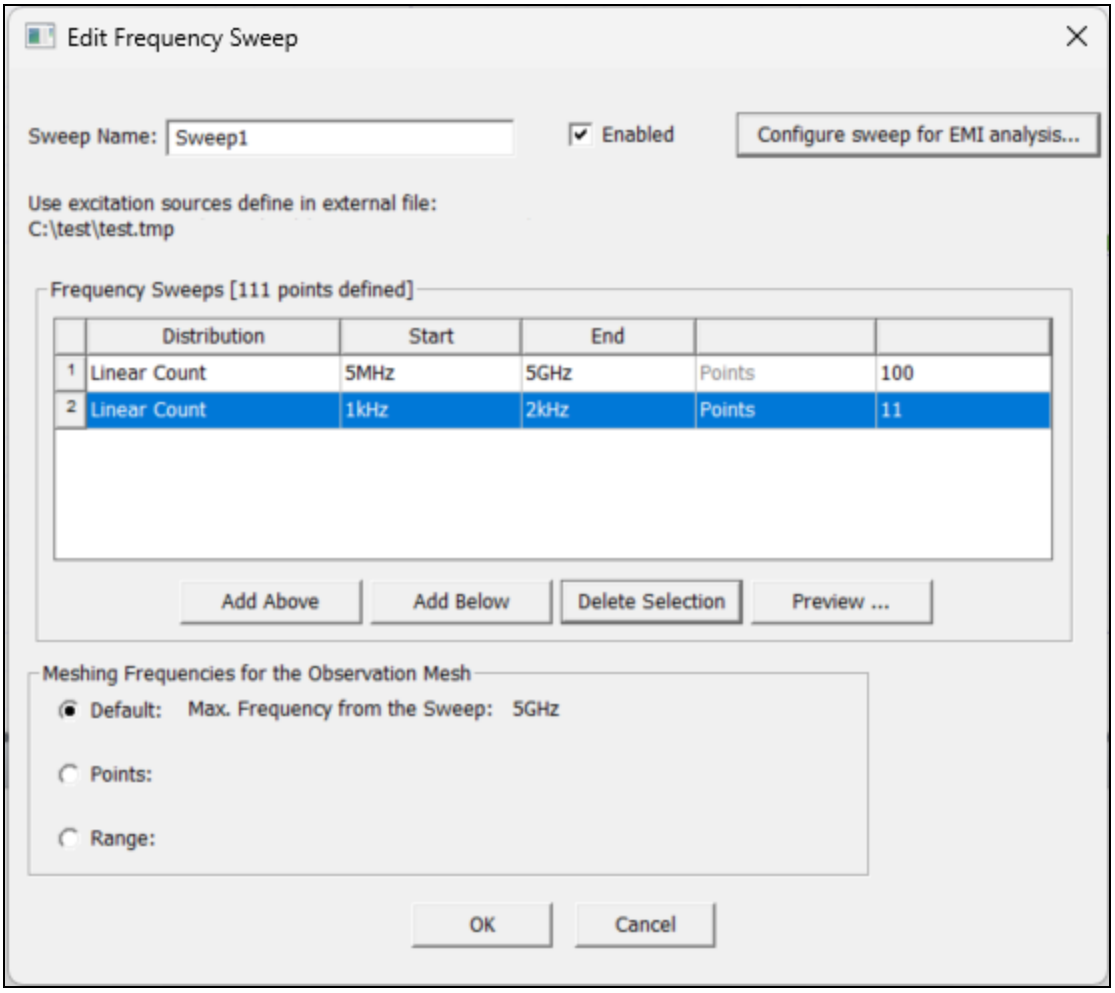
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Start Frequency** — enter a value for the minimum frequency to sweep (e.g., **1k**).
- **Stop Frequency** — enter a value for the maximum frequency to sweep (e.g., **2k**).
- **Calculate Optimal Sweep** — generates the sweeps. They appear as a list in the field.
- **Ignore transient stimuli before** — enter a value to filter out any non-periodic start-up transient ripple when computing the Fourier transform during simulation (i.e., **.8**).

> **Note:**
>
> Setting a value in the **Ignore transient stimuli before** box resets the X minimal to the entered value. The ignored time period is represented by a shaded rectangle on the plot.

2. Click **Apply Sweep** to add a new sweep row to the table in the **Frequency Sweeps** area in the **Edit Frequency Sweeps** window.
3. Click **Close** to return to the **Edit Frequency Sweeps** window. From the following example, a second row has been added with the default parameters.

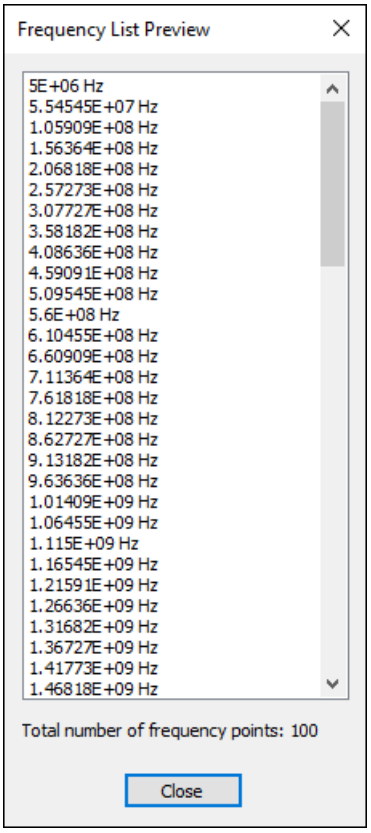
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Back at the **Edit Frequency Sweep** window, make selections from the following default values in the **Frequency Sweeps** area:
    - **Distribution** — select **Linear** or **By Decade** from the drop-down menu.
        - **Linear** — the difference between start frequency and stop frequency is calculated and divided by the number of solution points.
        - **By Decade** — distributes the number of points specified logarithmically, over each decade.
    - Enter values in the **Start**, **End**, and **Points** fields, as appropriate.
    - If applicable, click **Add Above** or **Add Below** to add additional frequency sweeps, including mixed sweep types.
    - If applicable, click **Delete** to remove any sweeps.

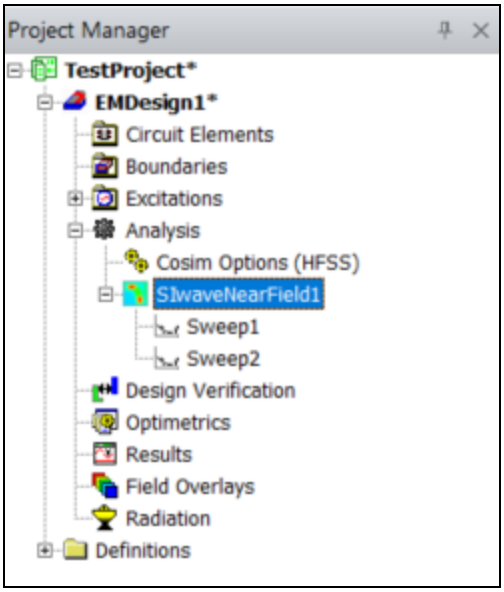
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Preview** to open the **Frequency List Preview** window. Then click **Close**.



5. From the **Meshing Frequencies for the Observation Mesh** area, select **Default**, **Points**, or **Range**.
    - **Default** — uses the default maximum frequency from the sweep (i.e., **5Ghz**).
    - **Points** — enter one or more specific frequencies to mesh.
    - **Range** — enter **Start**/**Stop** values to create a range to mesh.
6. Click **OK** and the sweep(s) will appear in the **Project Manager** window > **Project Tree** (e.g., **TestProject**) > active design folder > **Analysis** directory under the active 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.



Assigning a Transient Excitation

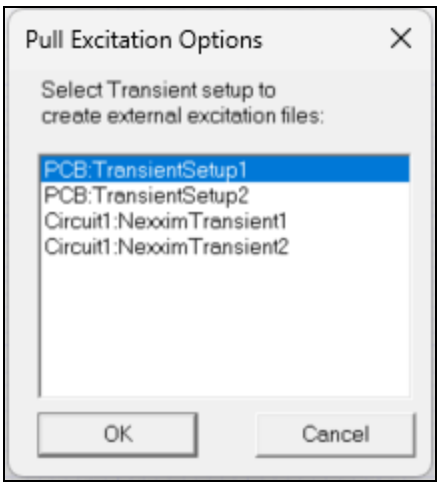
Follow these steps to select a Circuit or HFSS 3D Layout transient excitation source to use as an external source for simulation. These instructions assume the user has already set up and solved a Transient Analysis on the design with all appropriate drivers and receivers attached to ports. Transient currents flowing into each port are captured and converted to frequency-domain sources during execution of near field simulations. Users can also record these steps using the **Record Script** function on the **Automation** tab, then run the resulting Python script to set the specified excitation source. Refer to Using a Python Script to Configure an SIwave Solution Setup.

1. From the **Project Manager** window, expand the active design folder (e.g., **PullExecNF**) > subdesign (e.g., **U3:PCB**) > **Analysis**. Then right-click the near field solution setup (e.g., **Near Field Setup 1**) and select **Pull Excitations** to open the **Pull Excitation Options**

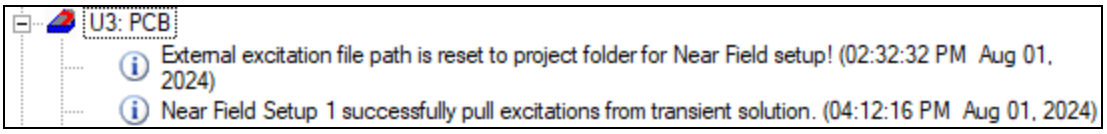
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

window.

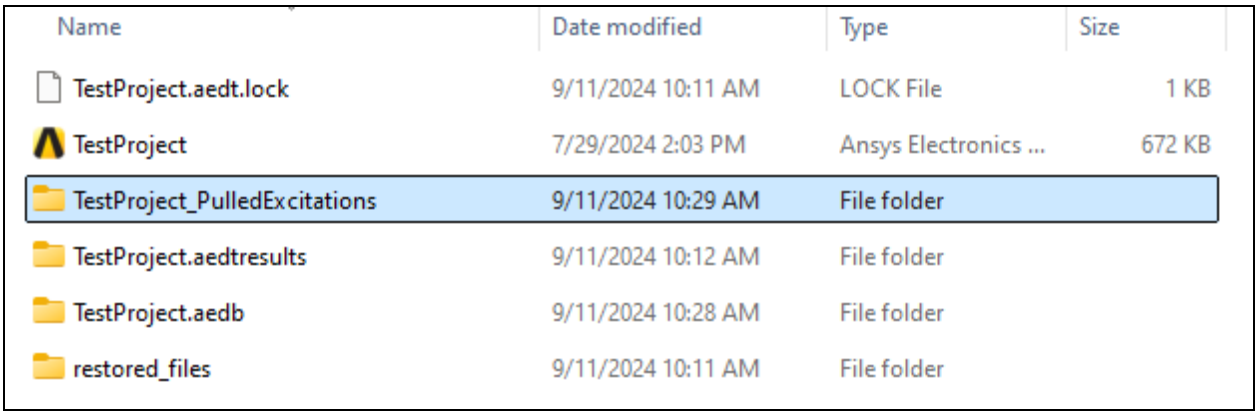
2. From the **Pull Excitation Options** window, select a design to use its excitations as an external source (e.g., **PCB:TransientSetup1**).



3. Click **OK** to close the **Pull Excitation Options** window. If the excitation is accepted, a confirmation message appears in the **Message Manager** window.

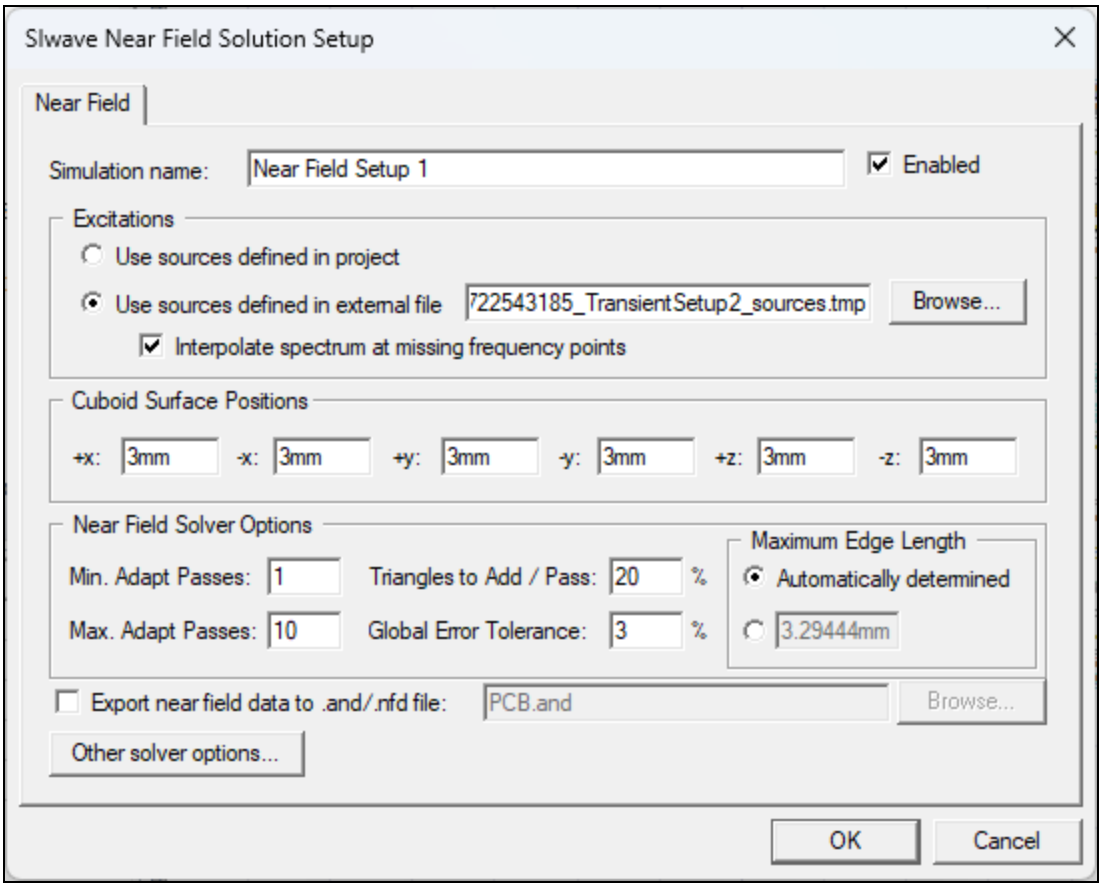


Within the same directory where the project is saved, a folder is created called ***<ProjectName>*_PulledExcitations**, which contains the newly created excitation files referred to in the **Message Manager** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **SIwave Near Field Solution Setup** window, **Use sources defined in external file** is now selected and the adjacent field is populated with the path to the excitations source file.

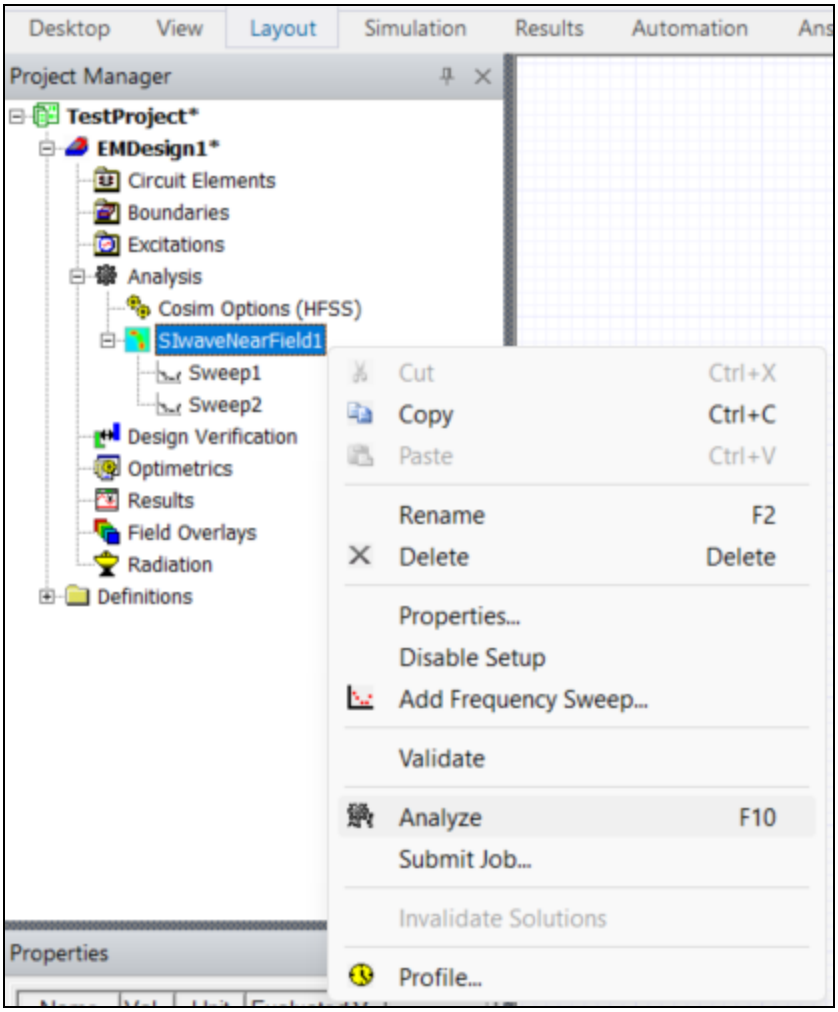


Analyzing the Design and Post-Processing

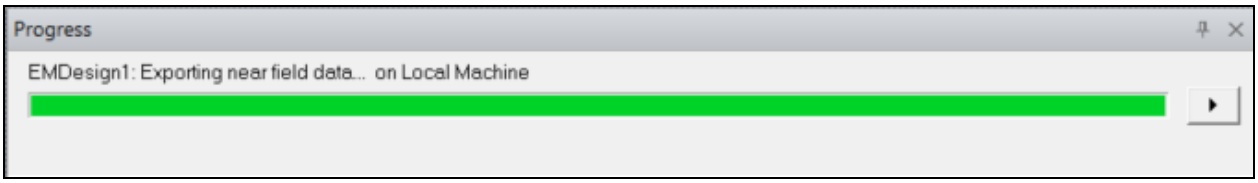
Follow these steps to analyze the new Near Field solution setup, then create an overlay, mesh, and generate a report.

1. From the **Project Manager** window, right-click the new simulation (e.g., **SIwaveNearField1**) and select **Analyze** to run the simulation. Refer to Running Simulations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

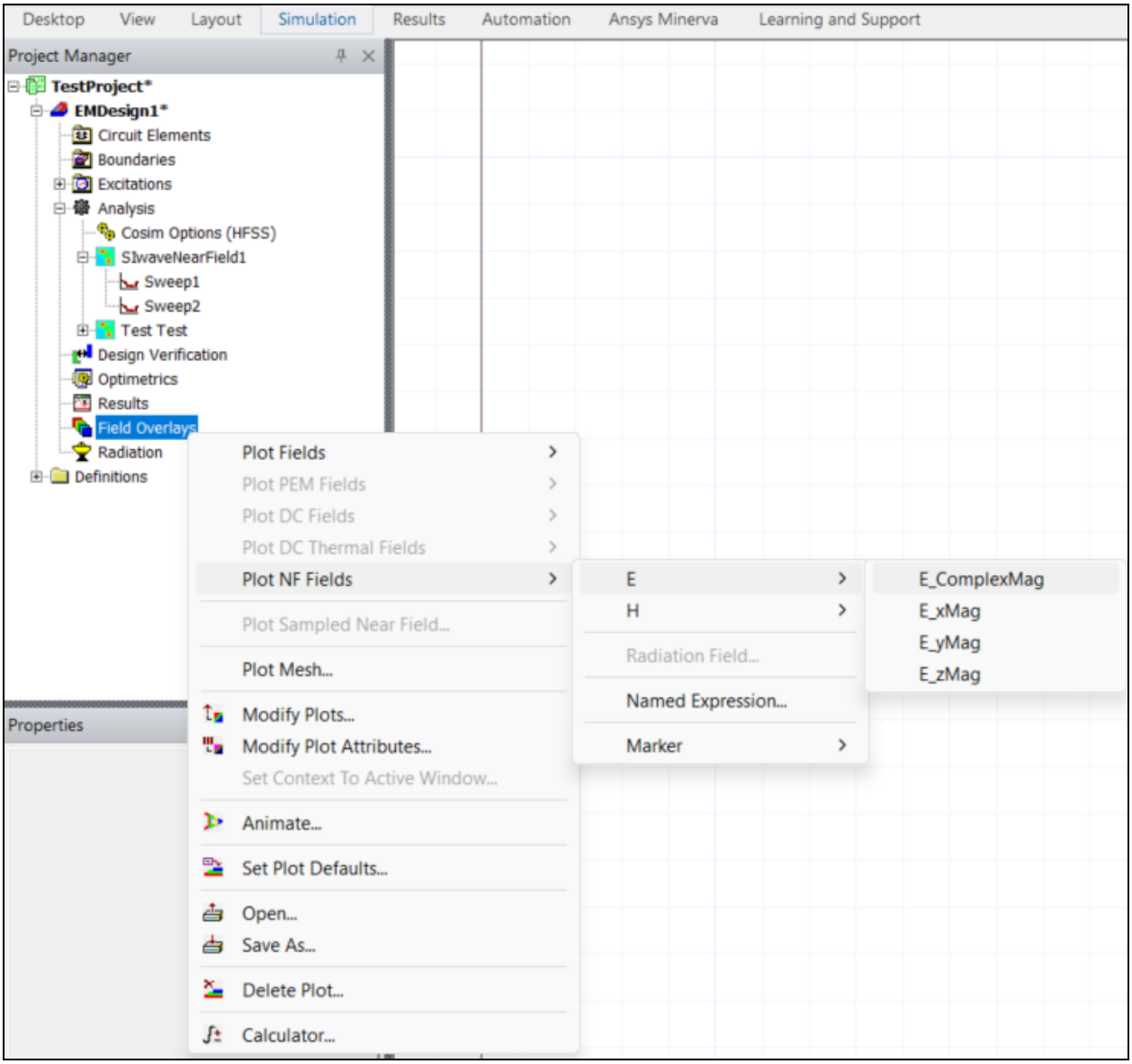


View progress in the **Progress** window.



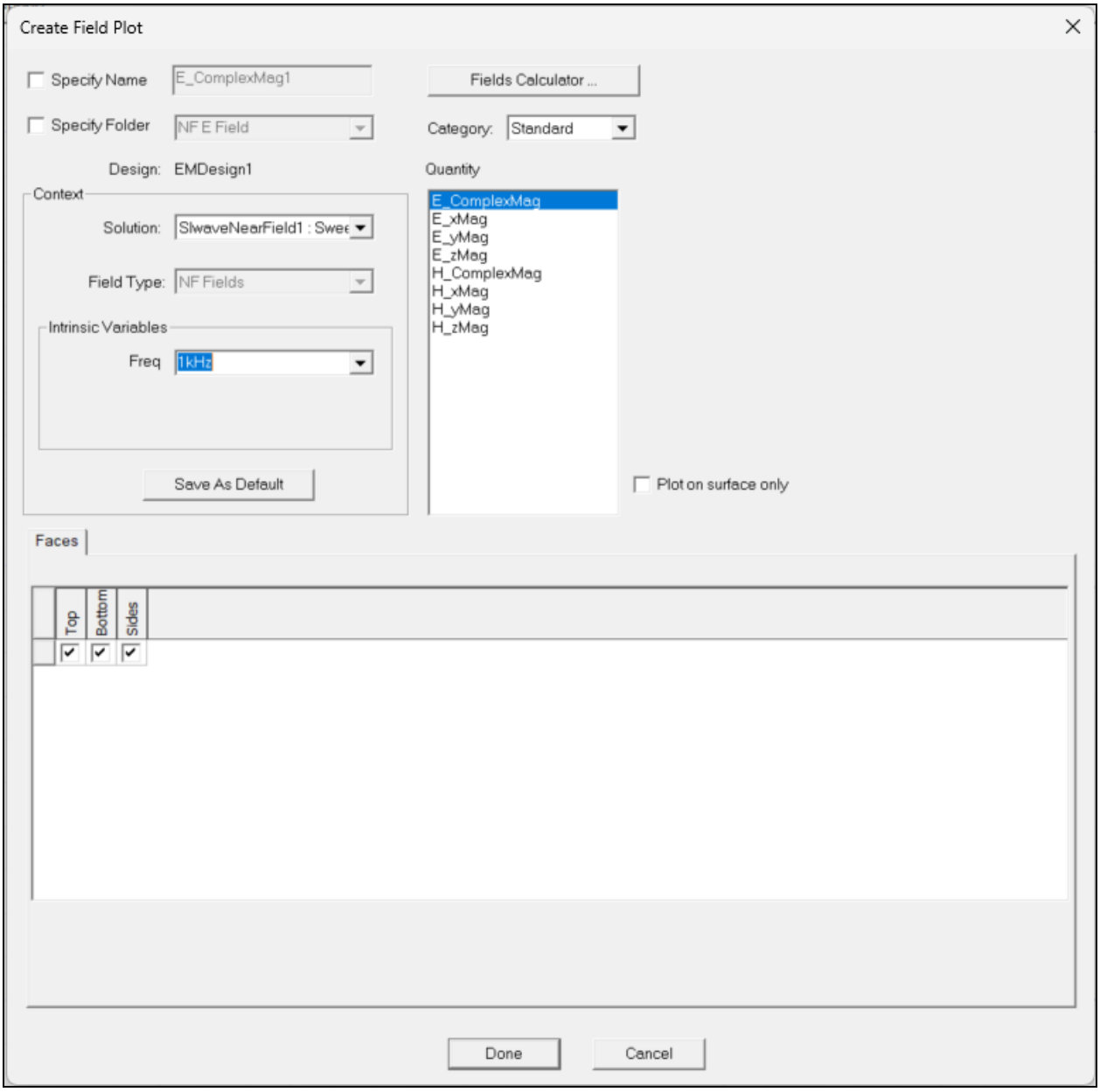
2. To create a field plot, navigate to the **Project Manager** window and right-click **Field Overlays**. Then select **Plot NF Fields** and any of the eight overlays available through **E** or **H** (e.g., **E** > **E_ComplexMag**) to open the **Create Field Plot** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

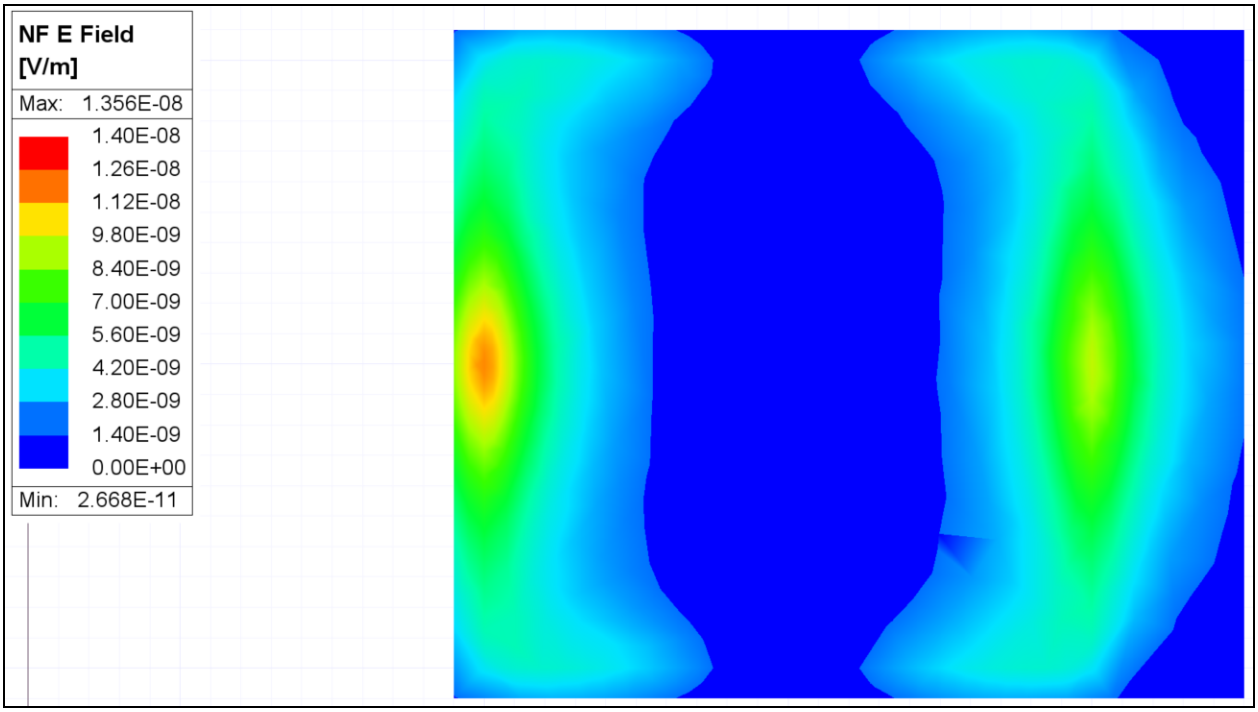


3. Select a **Quantity**, **Faces**, and any other appropriate options, then click **Done** to generate the field overlay. Refer to .

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

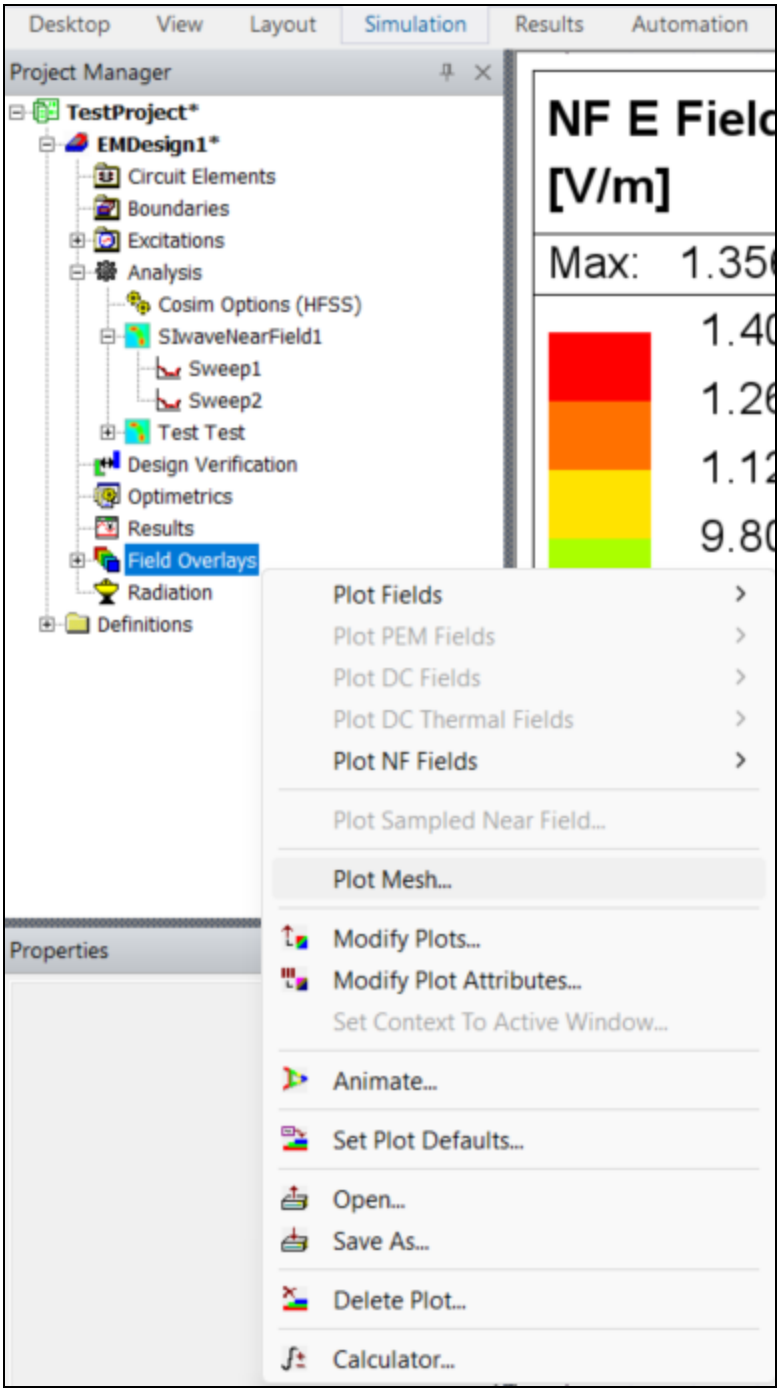


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



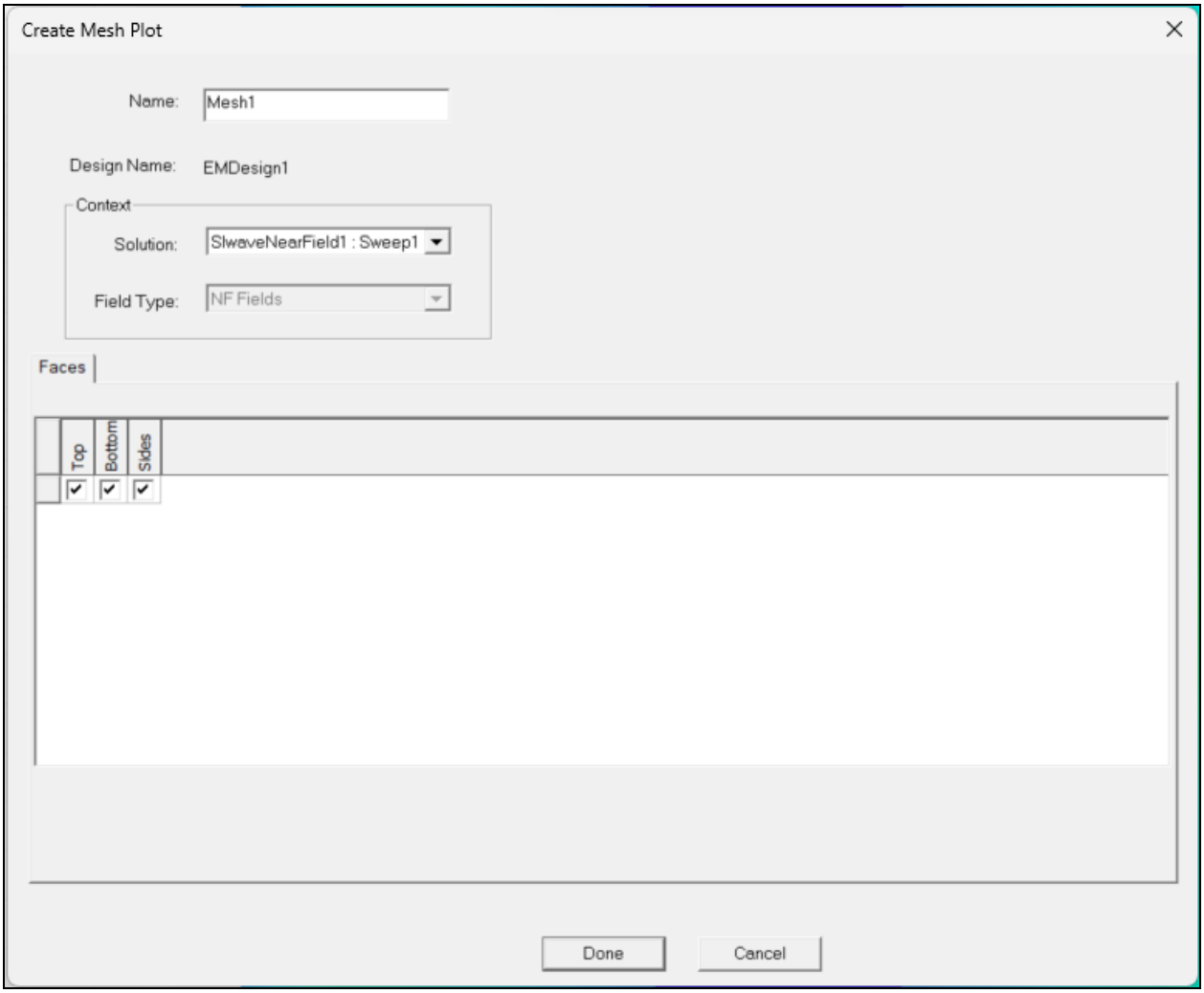
4. To create a mesh, navigate to the **Project Manager** window and right-click **Field Overlays** again. Then select **Plot Mesh** to open the **Create Mesh Plot** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

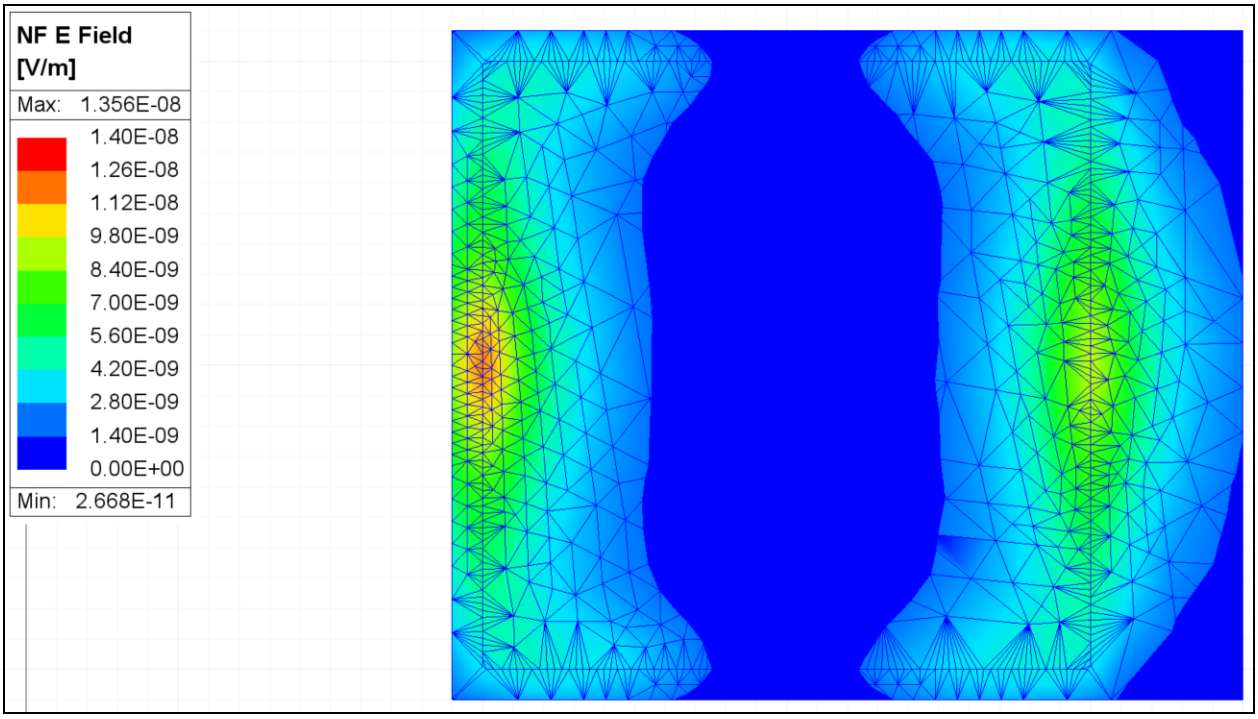


5. Select any/all **Faces** and any other appropriate options, then click **Done** to generate the mesh. Refer to .

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

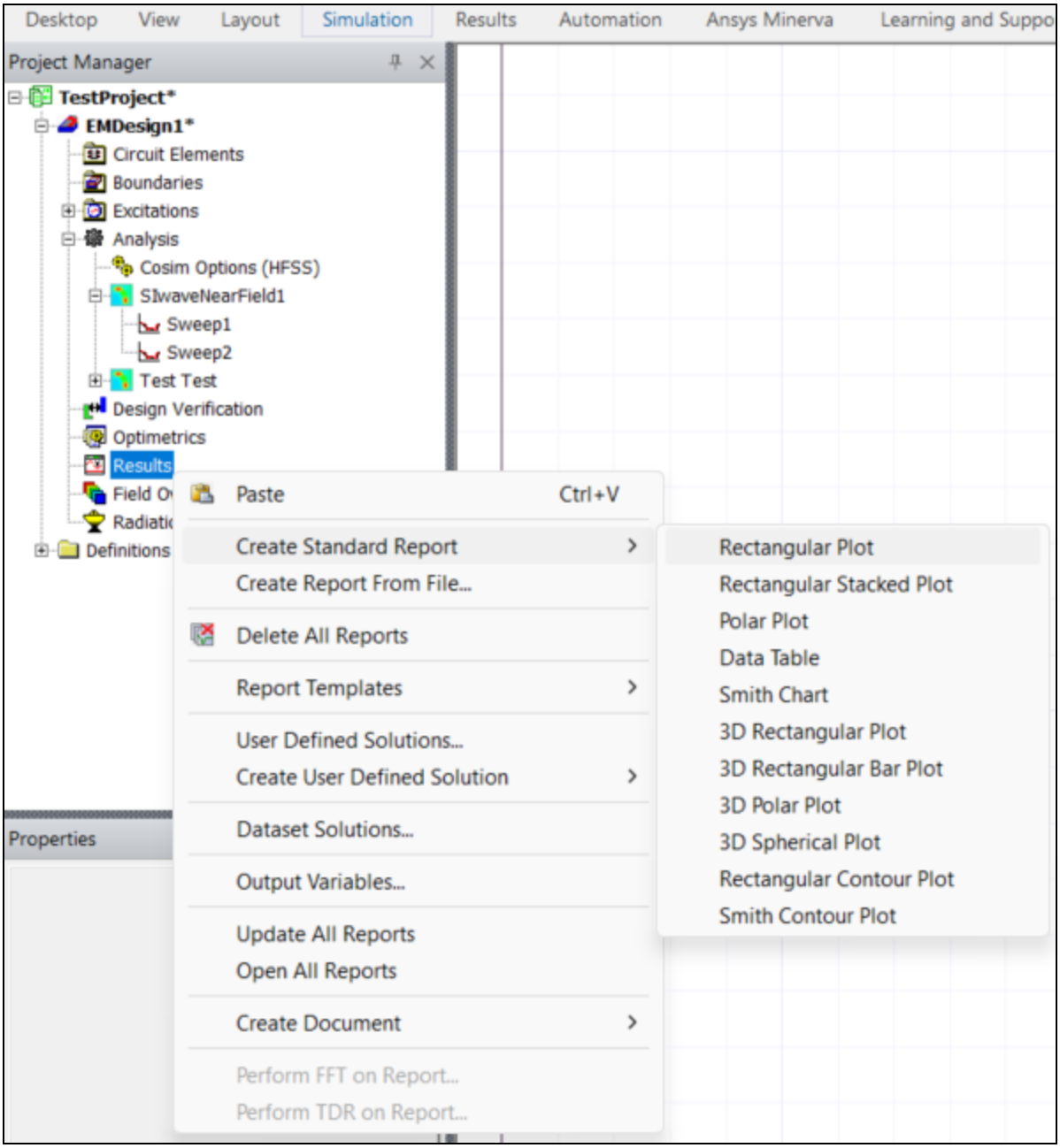


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



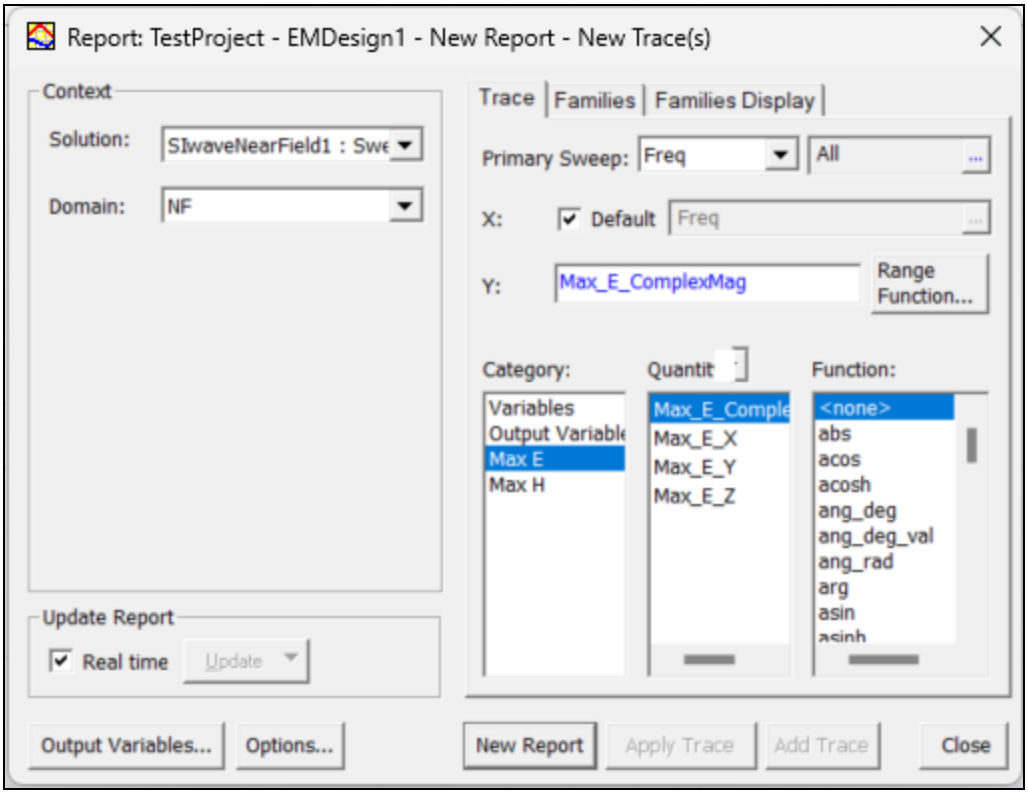
6. To create a report, navigate to the **Project Manager** window and right-click **Results**. Then navigate to **Create Standard Report** and select an appropriate report type (e.g., **Rectangular Plot**) to open the **Report** window.

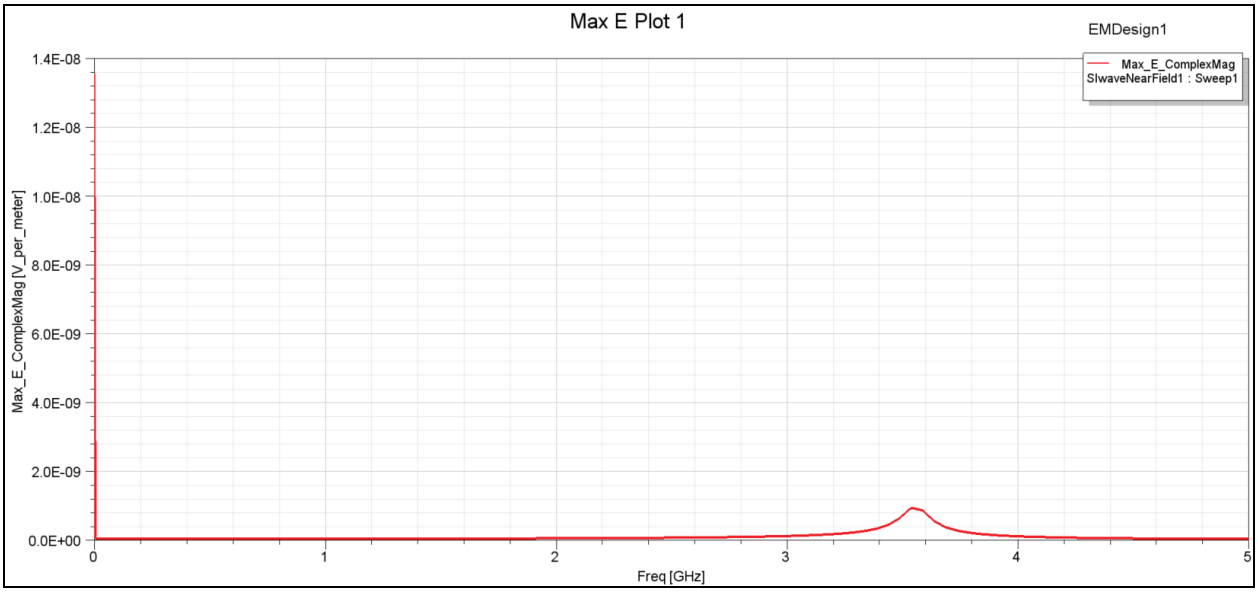
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Select a **Category**, any/all **Quantities**, and any other appropriate options (e.g., ensure **SIwaveNearField1 : Sweep1** is selected from the drop-down menu) and click **Done** to generate 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.





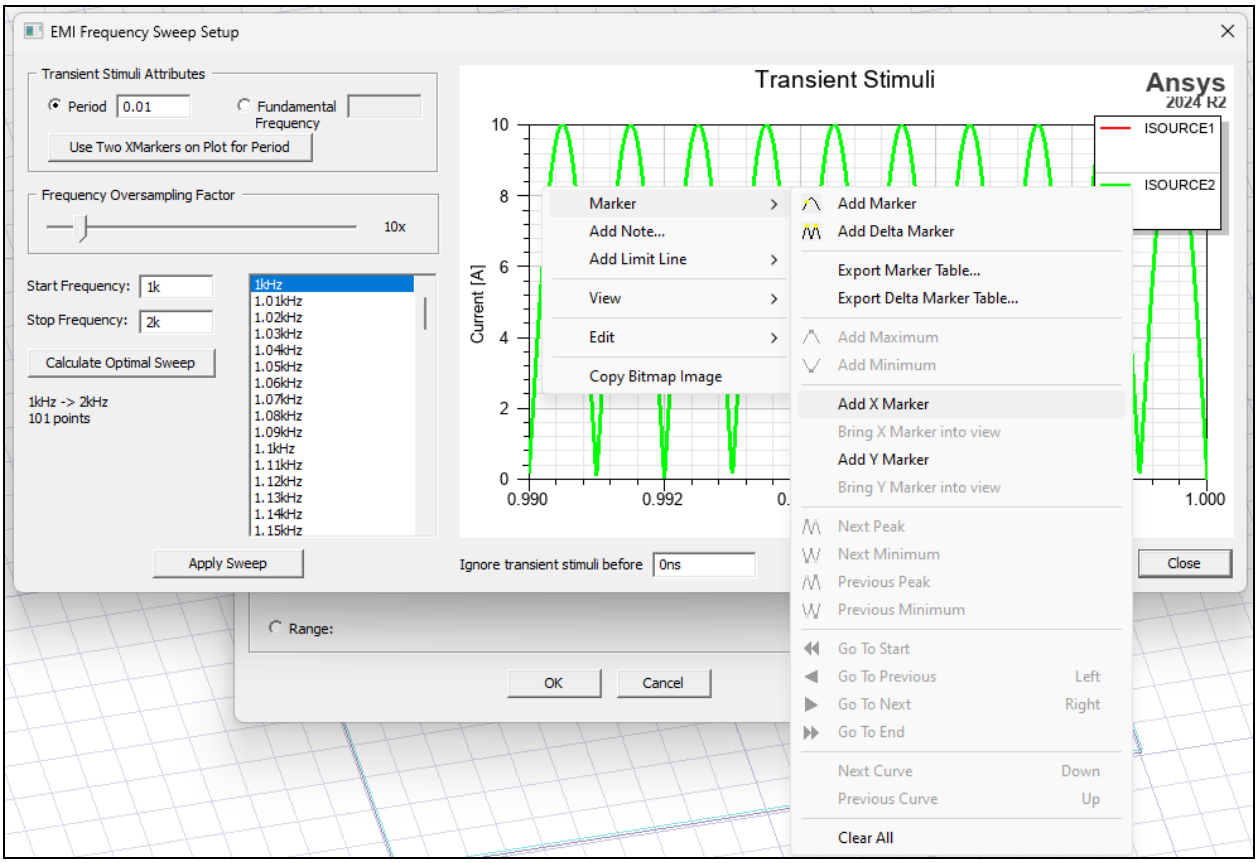
The procedure is complete.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Two X Markers

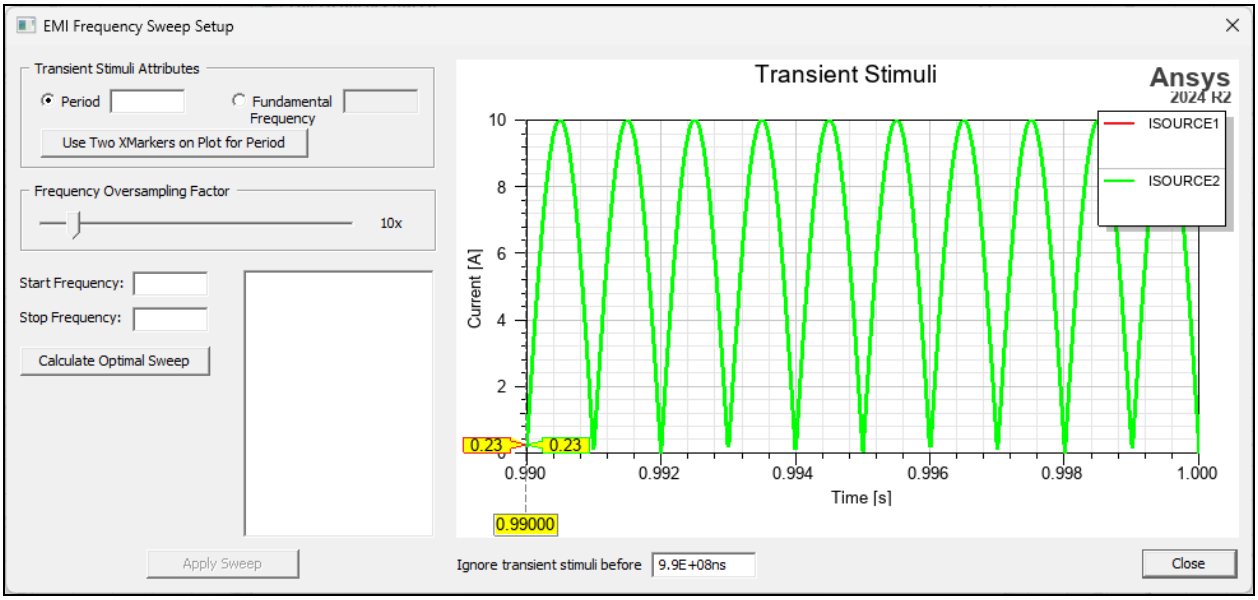
Follow these steps to mark two frequency points, then automatically determine the period between the frequency points.

1. Right-click within the graph and select **Add X Marker**.

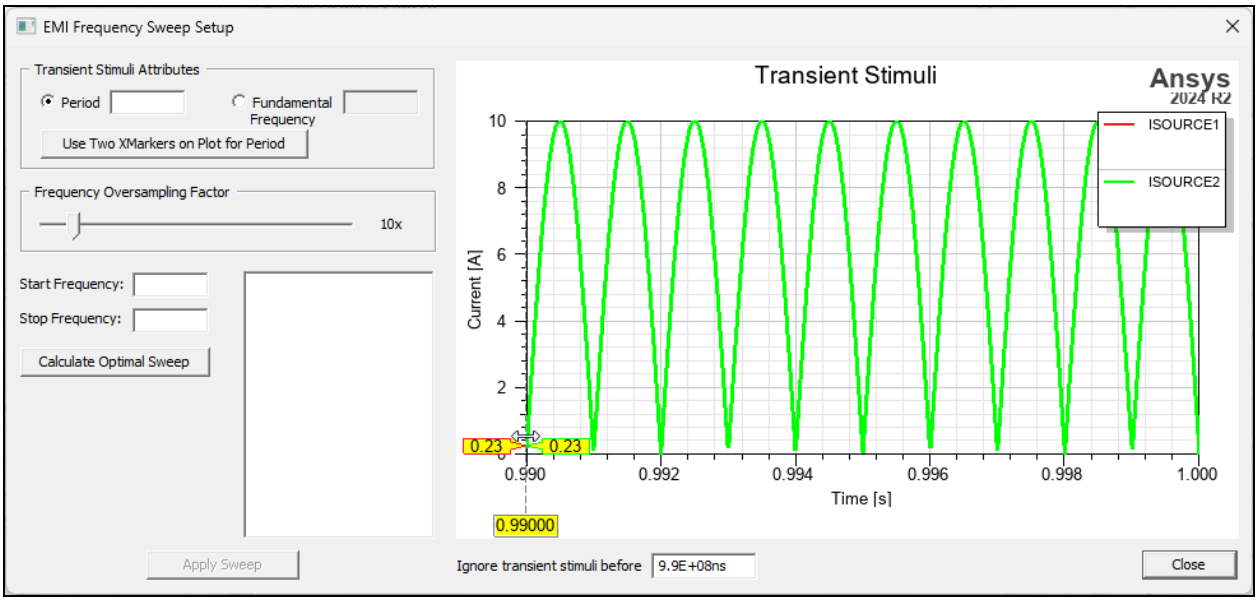


This creates the first X marker point (i.e., at X minimal, by default).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

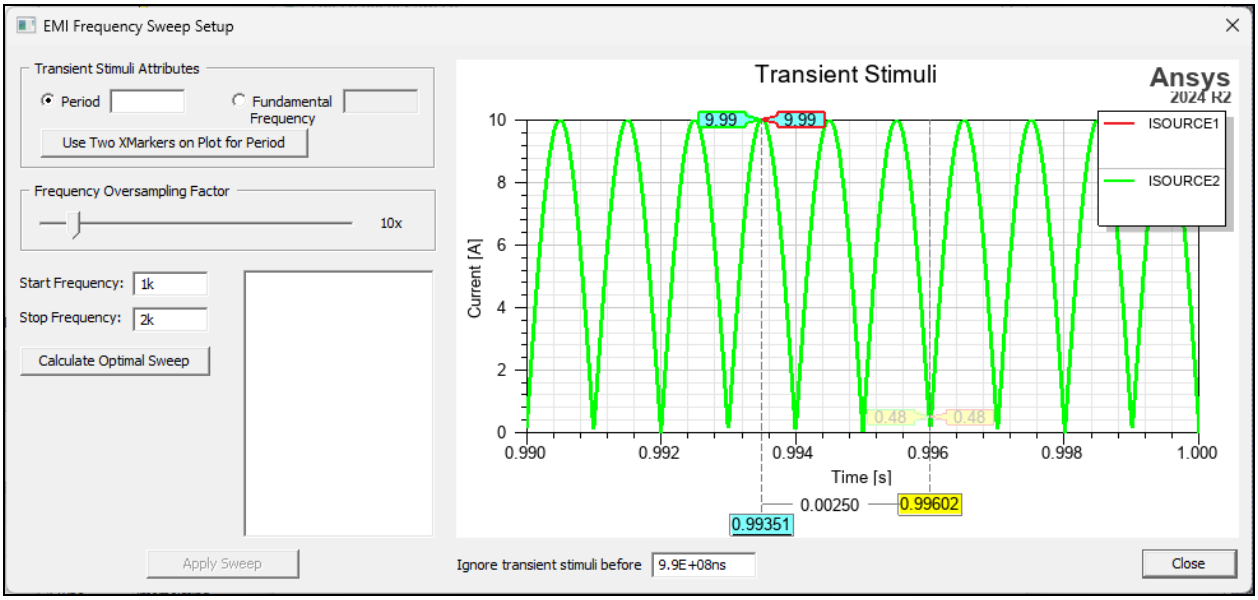


2. Hover directly over the X marker until the cursor becomes a line with two arrows. Then **click+drag** the X marker to the required frequency point.

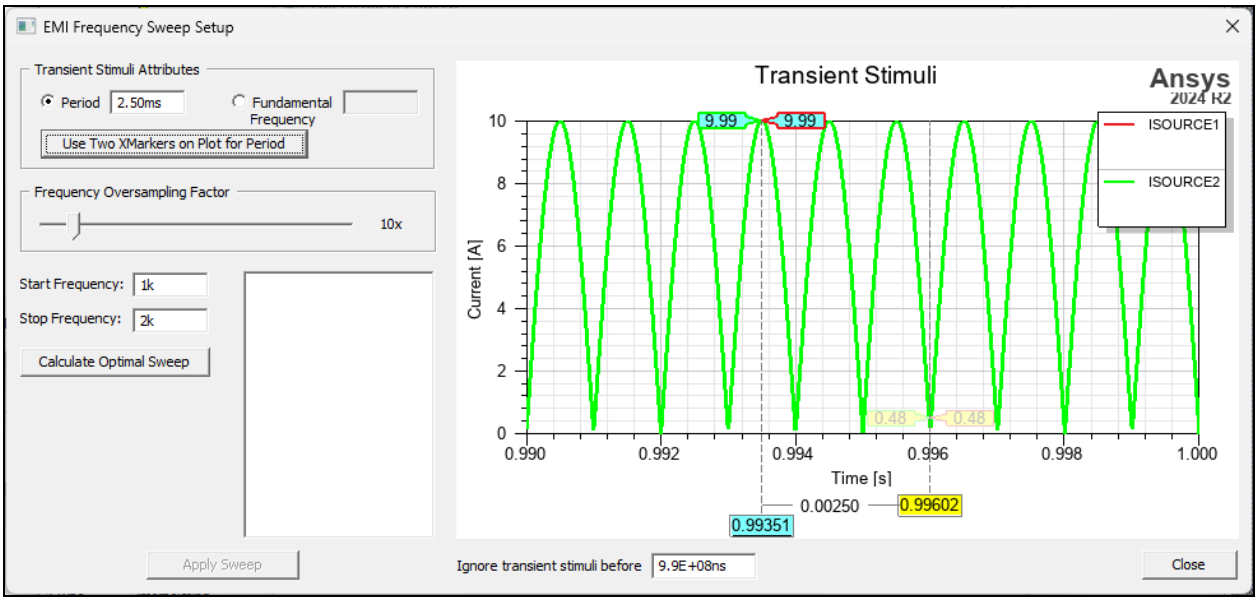


3. Right-click within the graph and select **Add X Marker** again to add a second X marker on the graphic.
4. **Click+drag** the second X marker to the required frequency point.

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 **Transient Stimuli Attributes** area, click **Use Two XMarkers on Plot for Period**. The **Period** box populates with the length between the user-selected frequency points (e.g., **2.50ms**).



Computing Far Field Solutions in HFSS 3D Layout (General Mode Only)

To analyze the radiated fields associated with a design, first define a radiation surface. The values of the fields calculated over this surface are used to compute the fields in the space

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surrounding the device. This space is typically split in two regions: near field and far field. The far field region exists at more than a wavelength from an energy source.

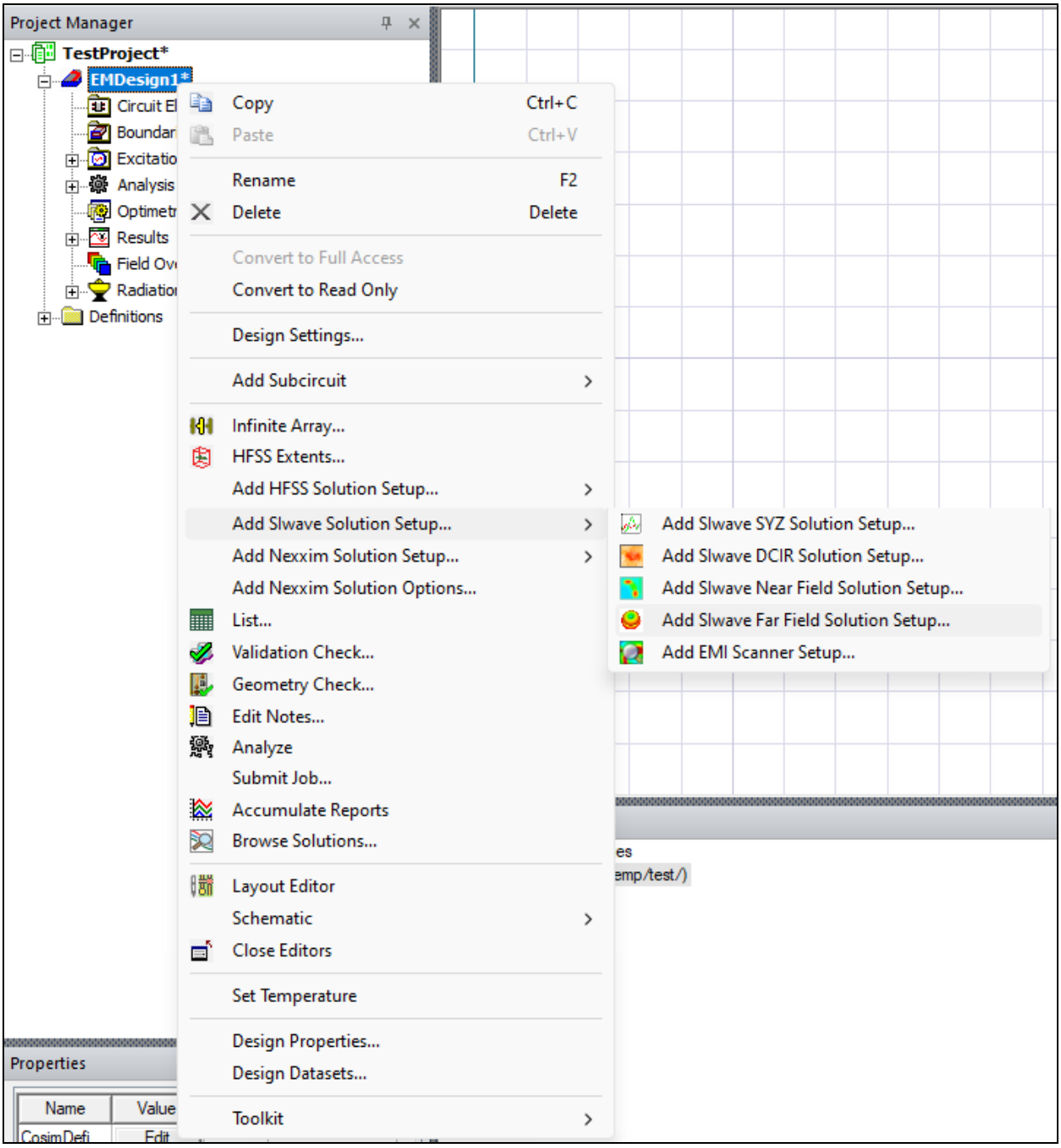
Setting Up a Far Field Simulation

Follow these steps to add a far field solution setup to an active design.

> **Note:**
>
> Users can also configure a far field solution setup using a Python script, either by manually creating one or utilizing the **Record Script** function on the **Automation** tab, then importing the result to generate an Ansys ECAD database file (i.e., **edb.def**) source. Refer to Using a Python Script to Configure an SIwave Solution Setup.

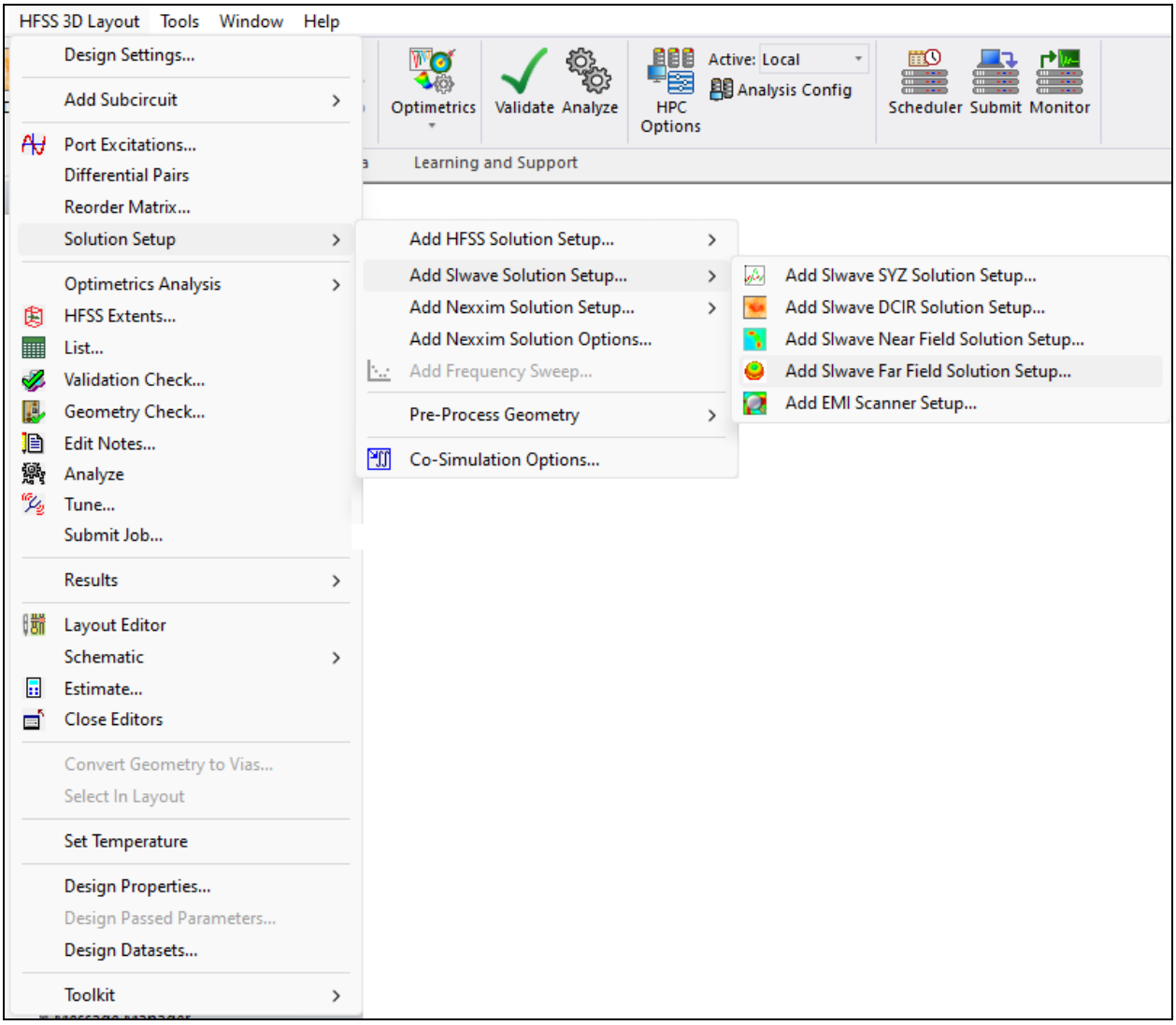
1. Open the **Far Field Solution Setup** window by doing one of the following:
    - From the **Project Manager** window, expand the **Project Tree** (e.g., **TestProject**). Then right-click the active design folder (e.g., **EMDesign1**) and select **Add SIwave Solution Setup** > **Add SIwave Far Field 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.

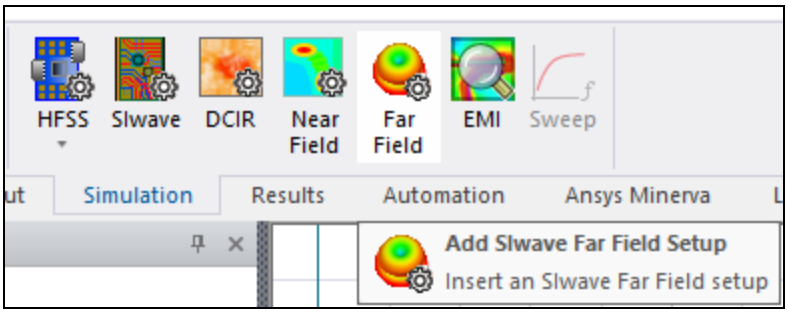


- From **HFSS 3D Layout**, select **Solution Setup** > **Add Slwave Solution Setup** > **Add Slwave Far Field Solution Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

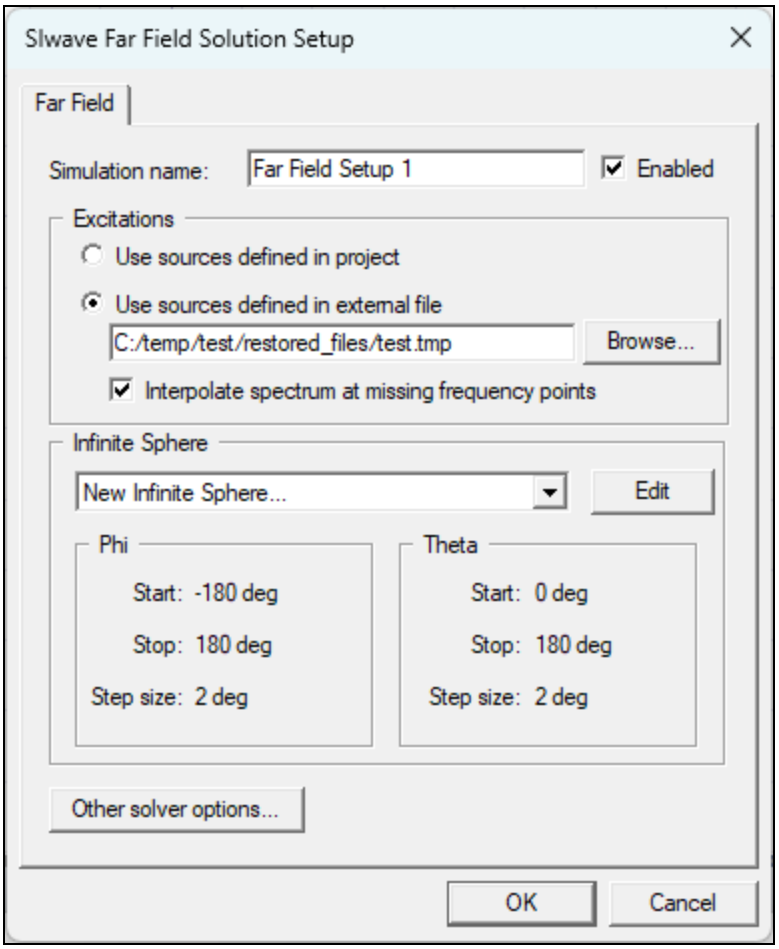


- From the **Simulation** ribbon tab, click **Far Field**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the **SIwave Far Field Solution Setup** window, make selections to change the following default values.



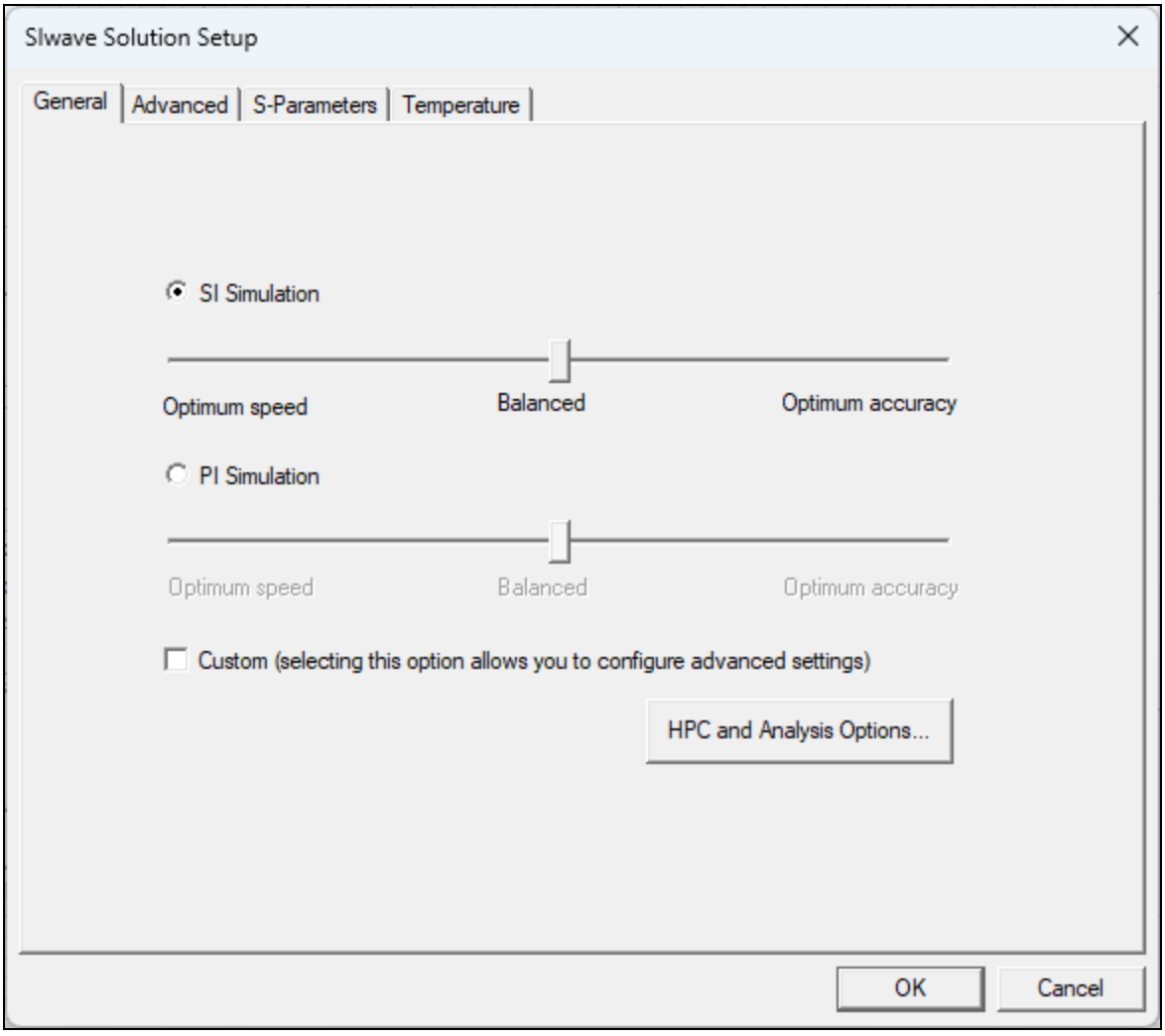
- **Simulation name** — enter a name in the field or accept the default (i.e., **Far Field Setup 1**).
- **Excitations** — select a source of excitations (i.e., **Use sources defined in the project** or **Use sources defined in external file**). If users select **Use source defined in external file**, **Browse** to an appropriate file and select it (e.g., **test.tmp**). If applicable, uncheck the **Interpolate spectrum at missing frequency points** 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:**
>
> A transient excitation source from either Circuit or HFSS 3D Layout sources can be assigned after setup is complete. Refer to Assigning a Transient Excitation.

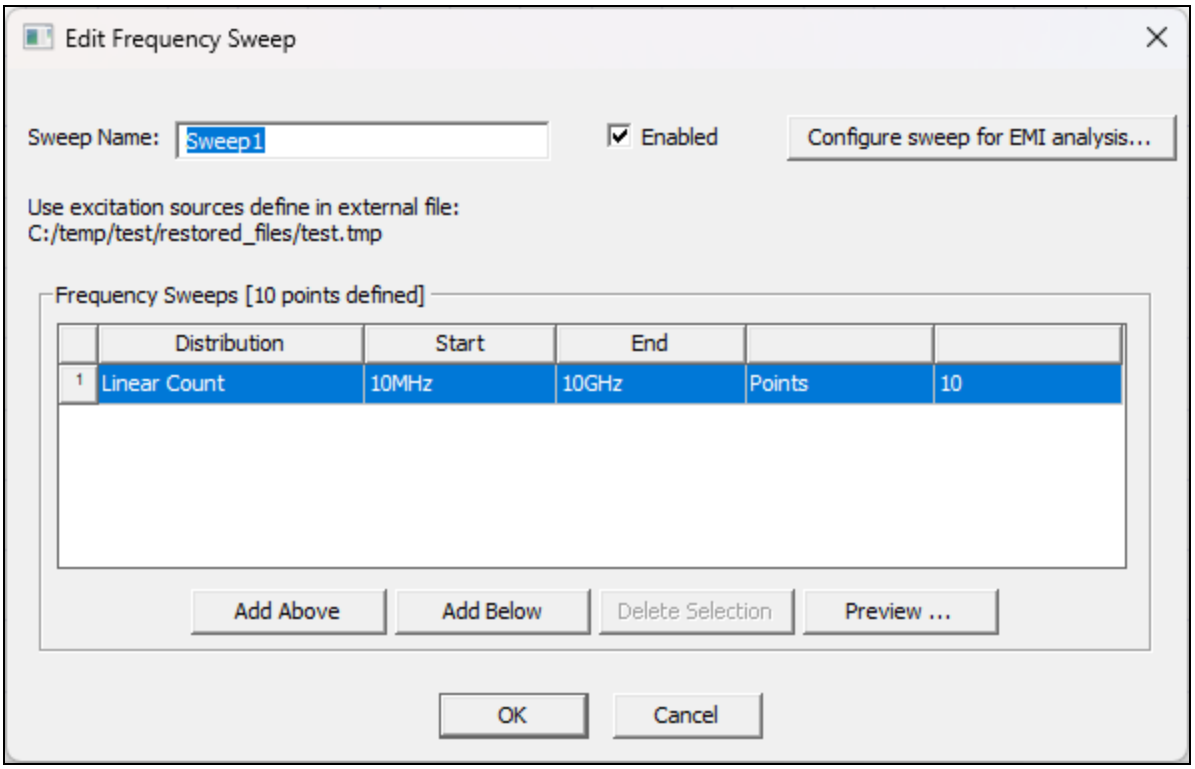
- **Infinite Sphere** — select a far field infinite sphere from the drop-down menu and click **Edit** to make changes to the selected infinite sphere. Alternatively, select **New Infinite Sphere** and click **Edit** to set up a new far field infinite sphere. Refer to Set Up a Far Field Infinite Sphere.

3. If applicable, click **Other solver options** to open the SIwave Solution Setup window. Otherwise, click **OK** to add the new far field solution setup 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.

The **Edit Frequency Sweep** window appears.



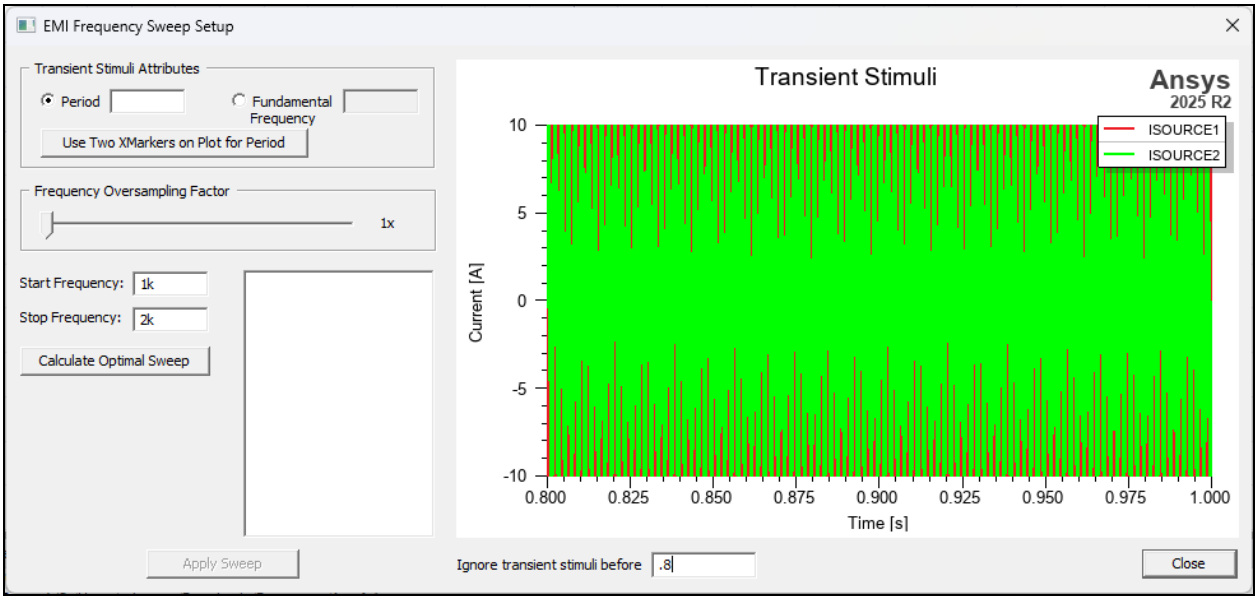
Setting Up a Frequency Sweep

Follow these steps to add a frequency sweep to a far field solution setup.

1. From the **Edit Frequency Setup** window, make selections from the following default values:
   - **Sweep Name** — enter a name in the field or accept the default (i.e., **Sweep1**).
   - **Enabled** — checked by default to enable the sweep.
   - **Configure sweep for EMI analysis** — click to access the **EMI Frequency Sweep Setup** window.

     > **Note:**
     >
     > EMI analysis is only available if the user selects **Use sources defined in external file** in the **Excitations** area of the **SIwave Far Field Solution Setup** window.

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 **EMI Frequency Sweep Setup** window, make selections from the following default values:
  - From the **Transient Stimuli Attributes** area, select **Period** or **Fundamental Frequency** and enter an appropriate value in the field (e.g., **0.01**). The period is the measured length of information (T). The Fundamental Frequency is defined as $f_0 = 1/T$ where T is the period information.

    > **Note:**
    >
    > To use the **Use Two XMarkers on Plot for Period** function to automatically determine the period, refer to .

  - **Frequency Oversampling Factor** – move the slider to select the frequency range sampling density (e.g., **1x**). A higher oversampling factor results in more frequency points being introduced between the selected sweep start and stop frequencies, and may result in longer simulation time.
  - **Start Frequency** — enter a value for the minimum frequency to sweep (e.g., **1k**).
  - **Stop Frequency** — enter a value for the maximum frequency to sweep (e.g., **2k**).
  - **Calculate Optimal Sweep** — generates the sweeps. They appear as a list in 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.

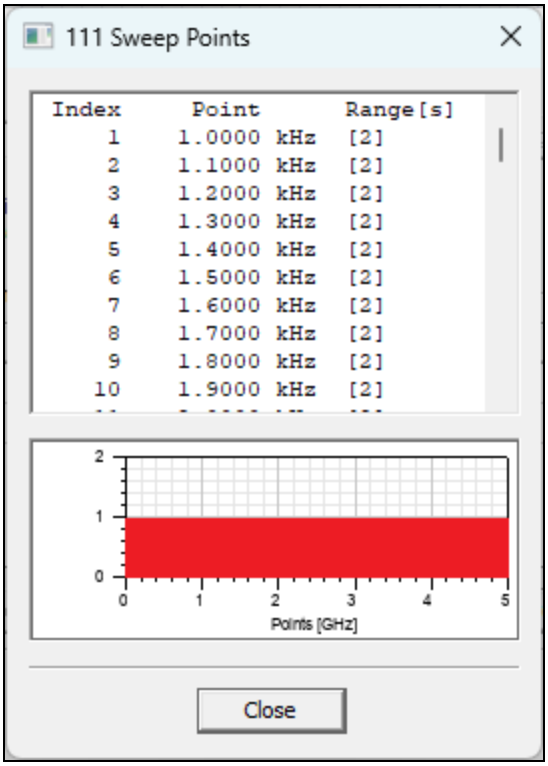
- **Ignore transient stimuli before** — enter a value to filter out any non-periodic start-up transient ripple when computing the Fourier transform during simulation (i.e., **.8**).

> **Note:**
>
> Setting a value in the **Ignore transient stimuli before** box resets the X minimal to the entered value. The ignored time period is represented by a shaded rectangle on the plot.

2. Click **Apply Sweep** to add a new sweep row to the table in the **Frequency Sweeps** area in the **Edit Frequency Sweeps** window.
3. Click **Close** to return to the **Edit Frequency Sweeps** window. Add more sweeps, as appropriate.
4. Back at the **Edit Frequency Sweep** window, make selections from the following default values in the **Frequency Sweeps** area:
    - **Distribution** — select **Linear** or **By Decade** from the drop-down menu.
        - **Linear** — the difference between start frequency and stop frequency is calculated and divided by the number of solution points.
        - **By Decade** — distributes the number of points specified logarithmically, over each decade.
    - Enter values in the **Start**, **End**, and **Points** fields, as appropriate.
    - If applicable, click **Add Above** or **Add Below** to add additional frequency sweeps, including mixed sweep types.
    - If applicable, click **Delete** to remove any sweeps.

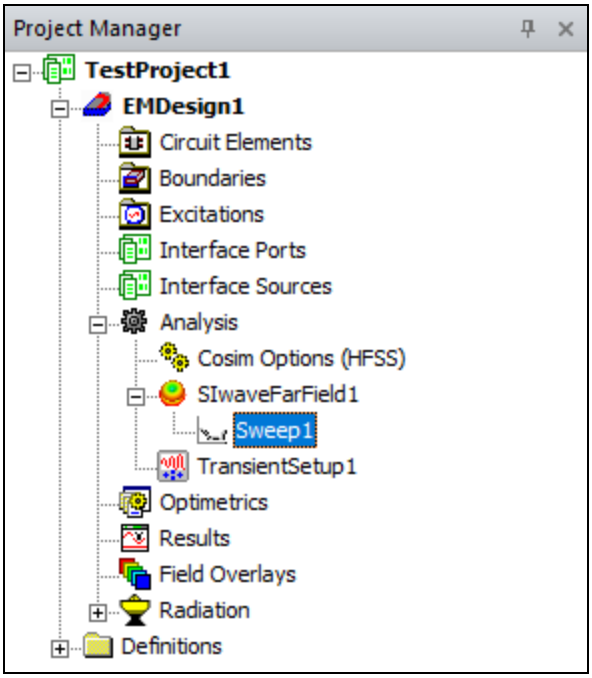
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Preview** to open the **Frequency List Preview** window. Then click **Close**.



5. Click **OK** and the sweep(s) will appear in the **Project Manager** window > **Project Tree** > active design folder > **Analysis** directory under the active 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.

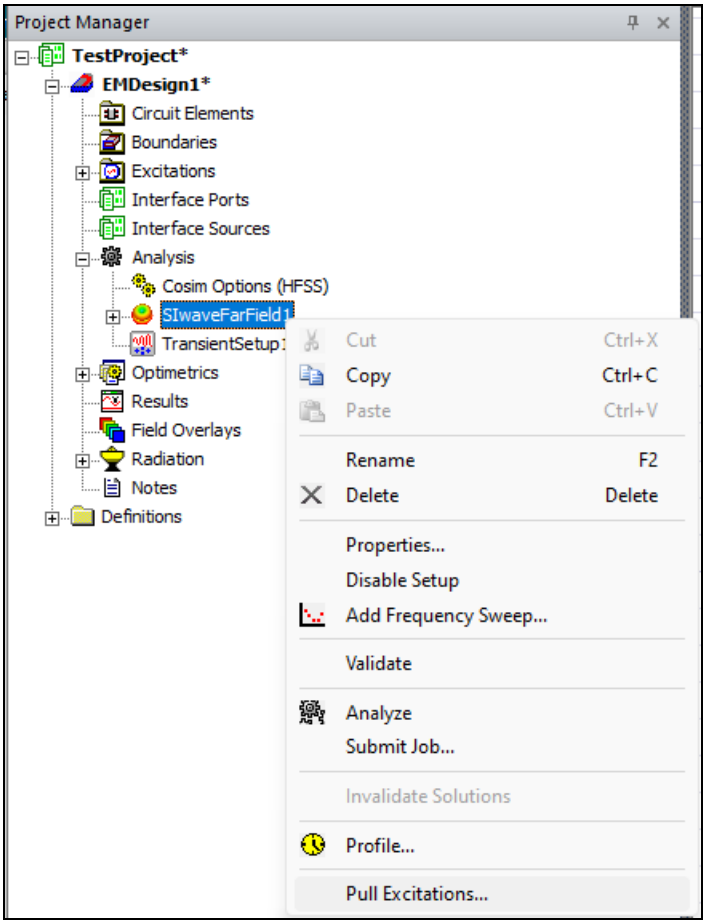


Assigning a Transient Excitation

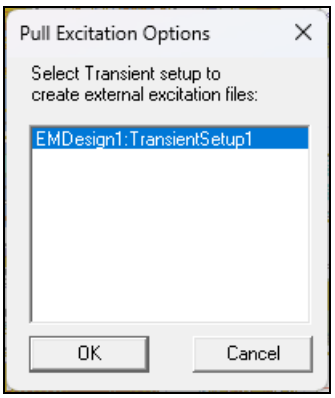
Follow these steps to select a Circuit or HFSS 3D Layout transient excitation source to use as an external source for simulation. These instructions assume the user has already set up and solved a Transient Analysis on the design with all appropriate drivers and receivers attached to ports. Transient currents flowing into each port are captured and converted to frequency-domain sources during execution of far field simulations. Users can also record these steps using the **Record Script** function on the **Automation** tab, then run the resulting Python script to set the specified excitation source. Refer to Using a Python Script to Configure an SIwave Solution Setup.

1. From the **Project Manager** window, expand the active design folder (e.g., **EMDesign1**) (and any subdesign) > **Analysis**. Then right-click the far field solution setup (e.g., **SIwaveFarField1**) and select **Pull Excitations** to open the **Pull Excitation 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.

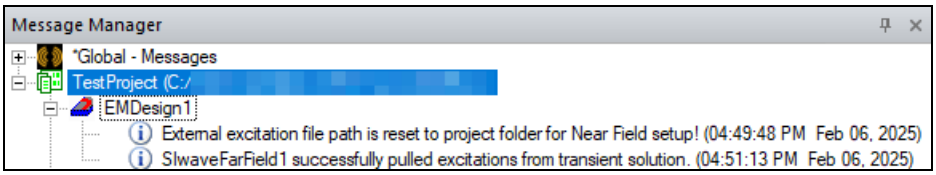


2. From the **Pull Excitation Options** window, select a design to use its excitations as an external source (e.g., **EMDesign1:TransientSetup1**).

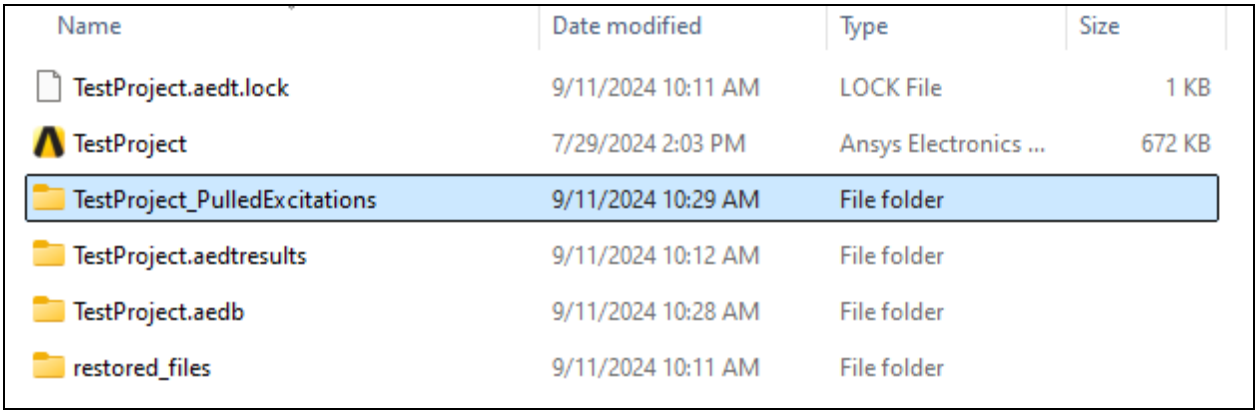


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK** to close the **Pull Excitation Options** window. If the excitation is accepted, a confirmation message appears in the **Message Manager** window.

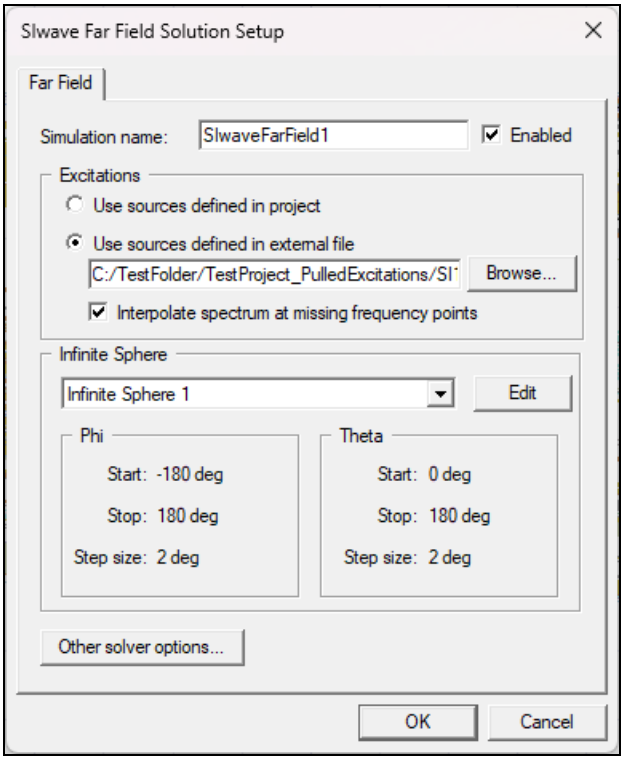


Within the same directory where the project is saved, a folder is created called ***<ProjectName>*_PulledExcitations**, which contains the newly created excitation files referred to in the **Message Manager** window.



From the **SIwave Far Field Solution Setup** window, **Use sources defined in external file** is now selected and the adjacent field is populated with the path to the excitations source file.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

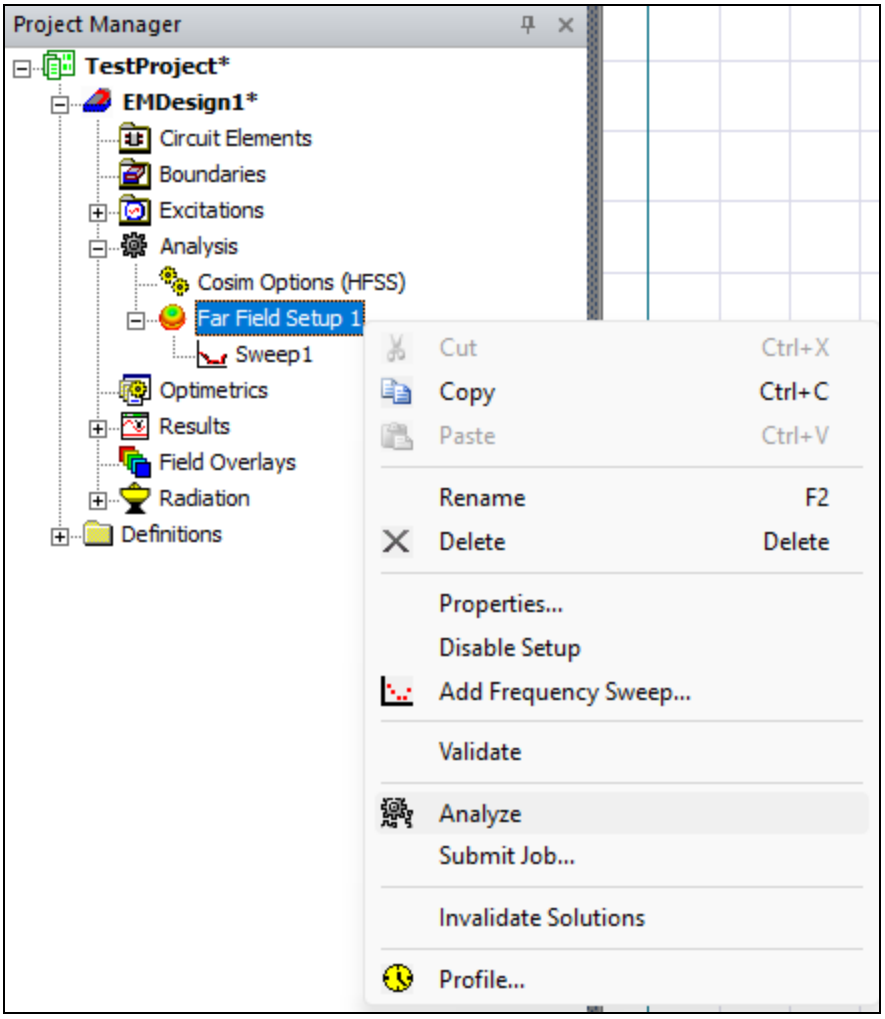


Analyzing the Design and Post-Processing

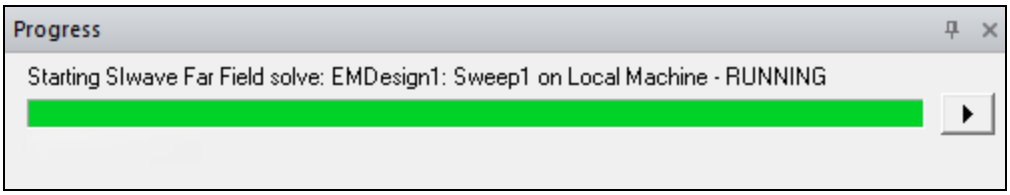
Follow these steps to analyze the new far field solution setup, then generate a report.

1. From the **Project Manager** window, right-click the new simulation (e.g., **Far Field Setup 1**) and select **Analyze** to run the simulation. Refer to Running Simulations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



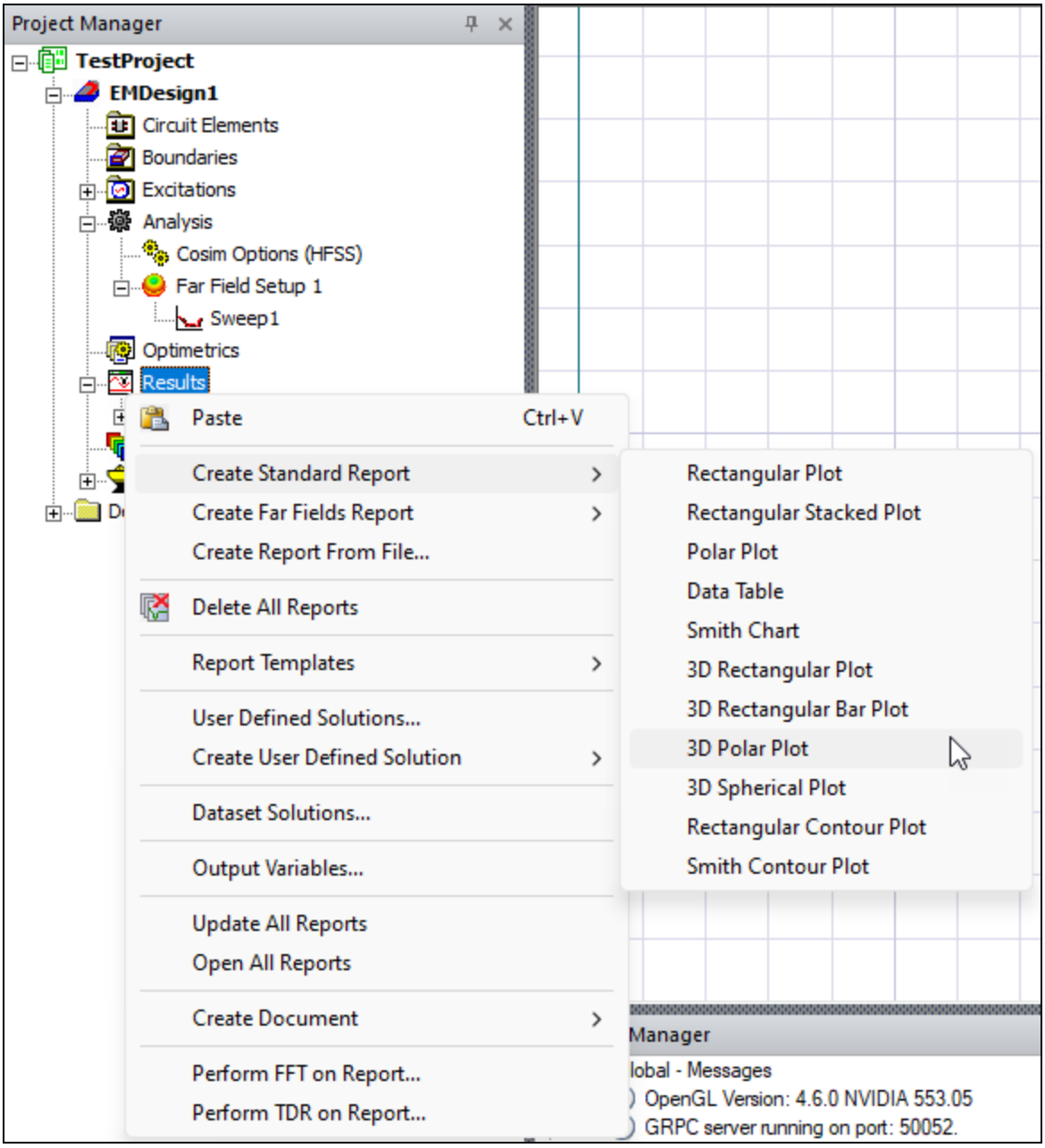
View progress in the **Progress** window.



The result of the analysis is saved in the same directory as the project file.

2. To create a report, such as a 3D polar plot report, navigate to the **Project Manager** window and right-click **Results**. Then navigate to **Create Standard Report** and select an appropriate report type (e.g., **3D Polar Plot**) to open the **Report** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



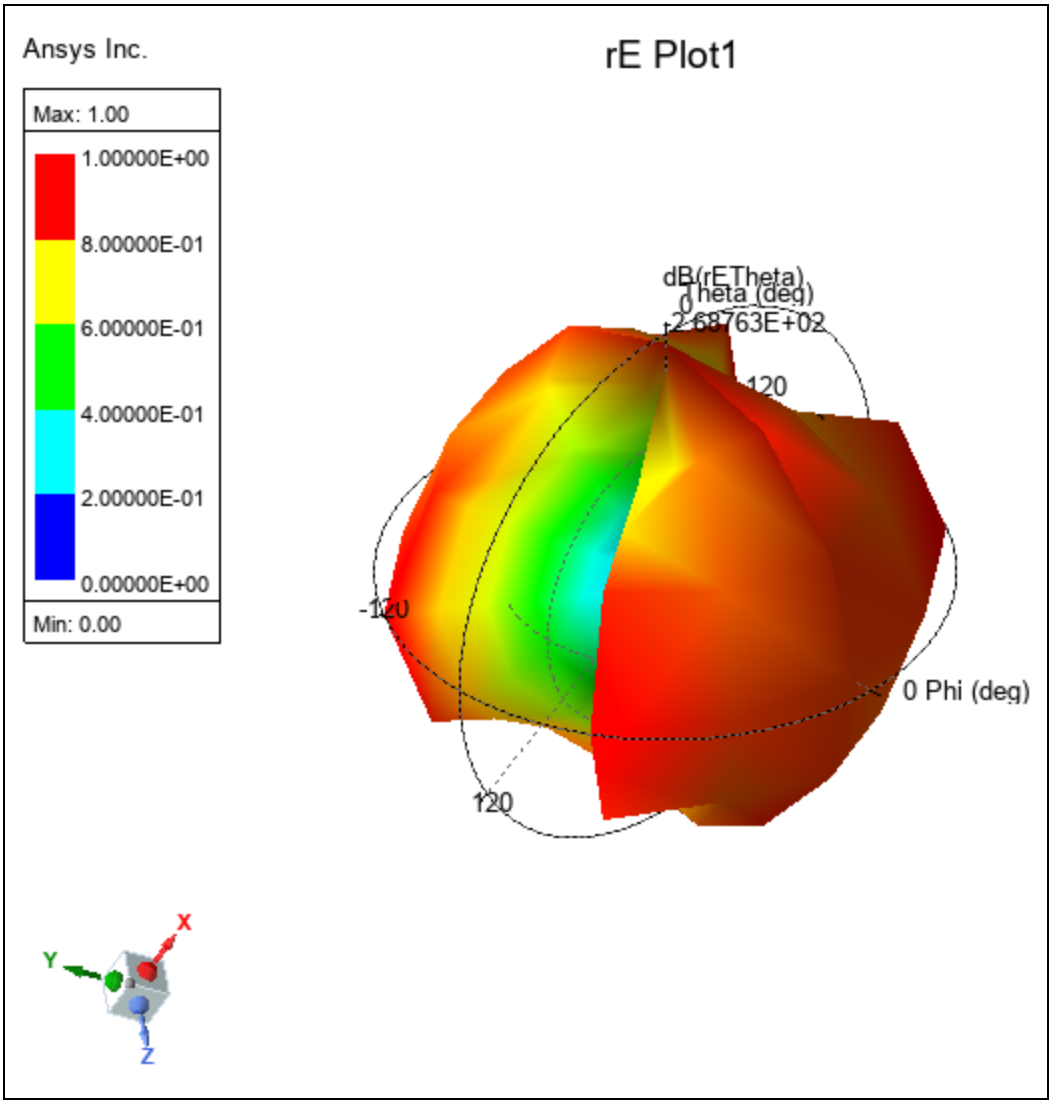
3. From the **Report** window, make any appropriate adjustments. For example:
   - Ensure the correct setup and sweep are selected from the **Solution** drop-down menu (e.g., **Far Field Setup 1 : Sweep1**).
   - Ensure the correct infinite sphere is selected in the **Geometry** drop-down menu

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(e.g., **Infinite Sphere 2**).

- **rE** is selected from the **Category** list.

4. Click **New Report** to generate the report. It will automatically open in the **View** tab.



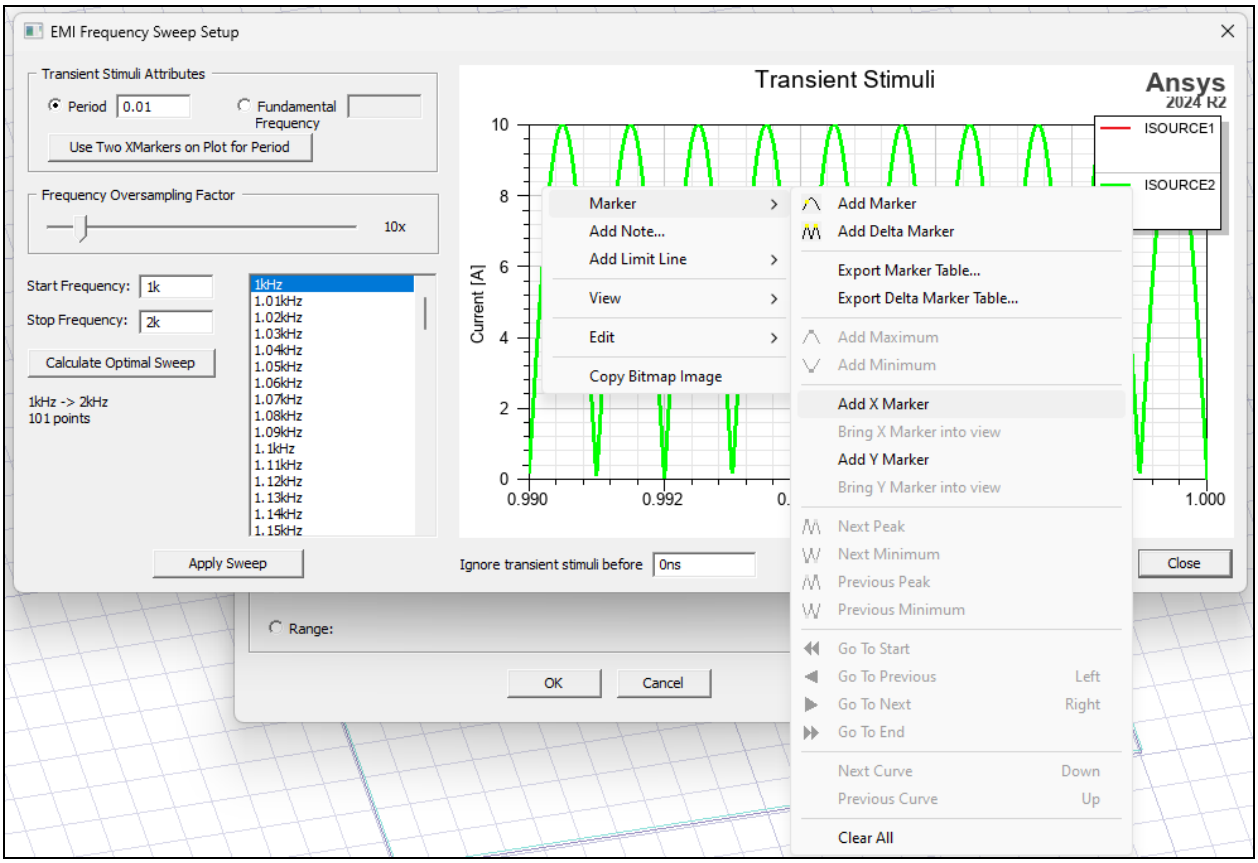
The procedure is complete.

Using Two X Markers

Follow these steps to mark two frequency points, then automatically determine the period between the frequency points.

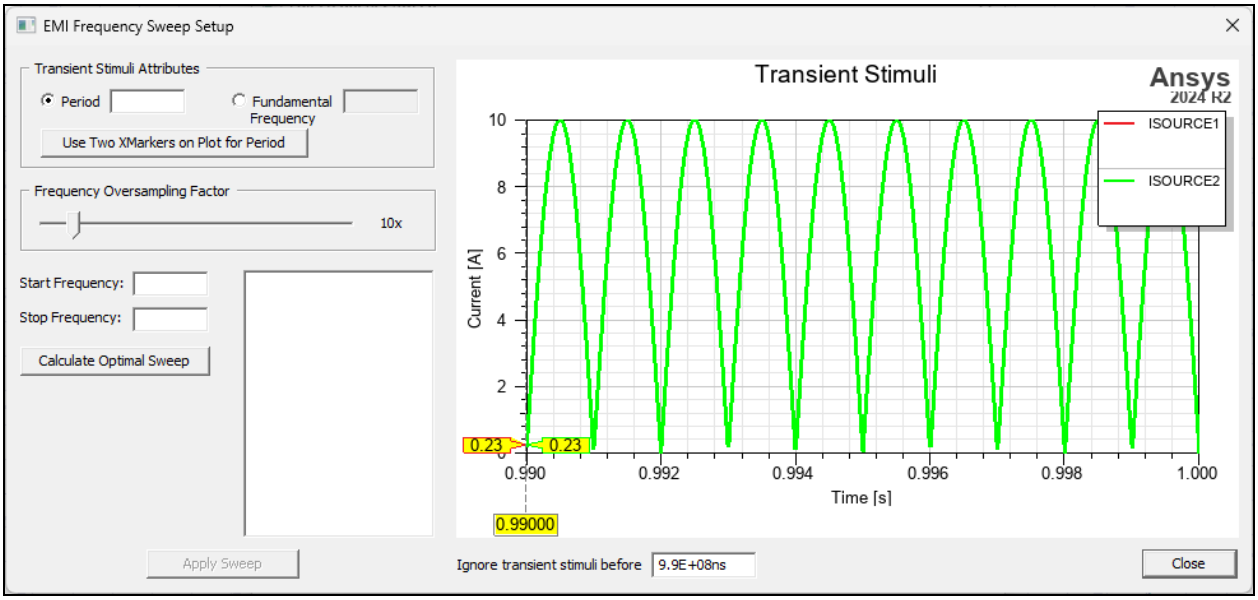
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click within the graph and select **Add X Marker**.

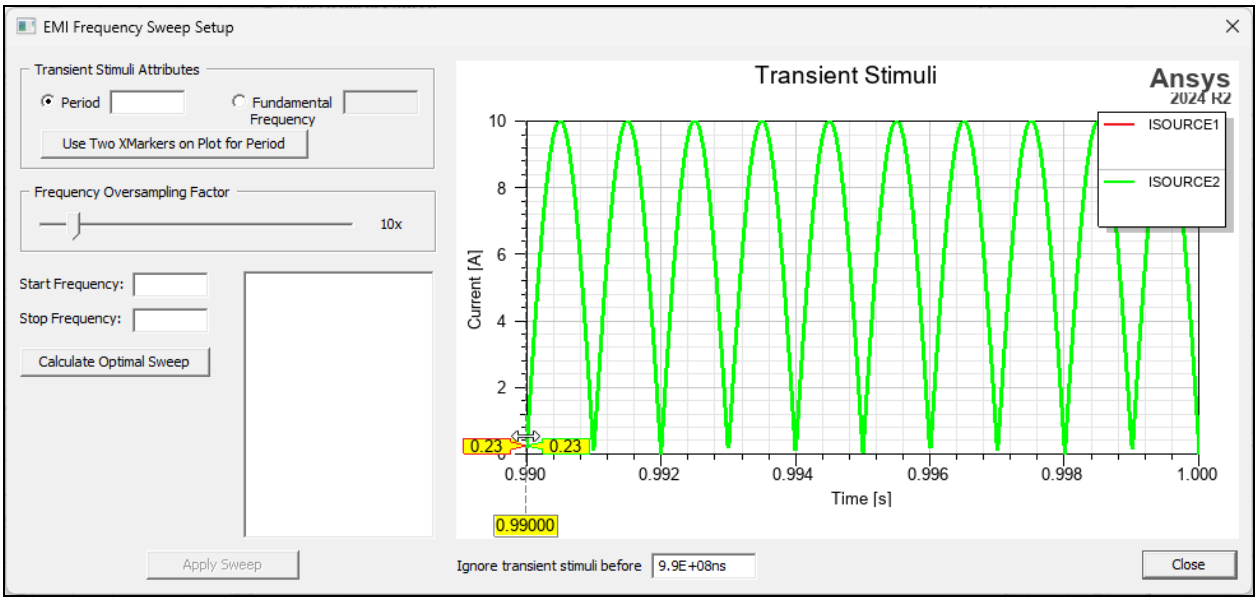


This creates the first X marker point (i.e., at X minimal, by default).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

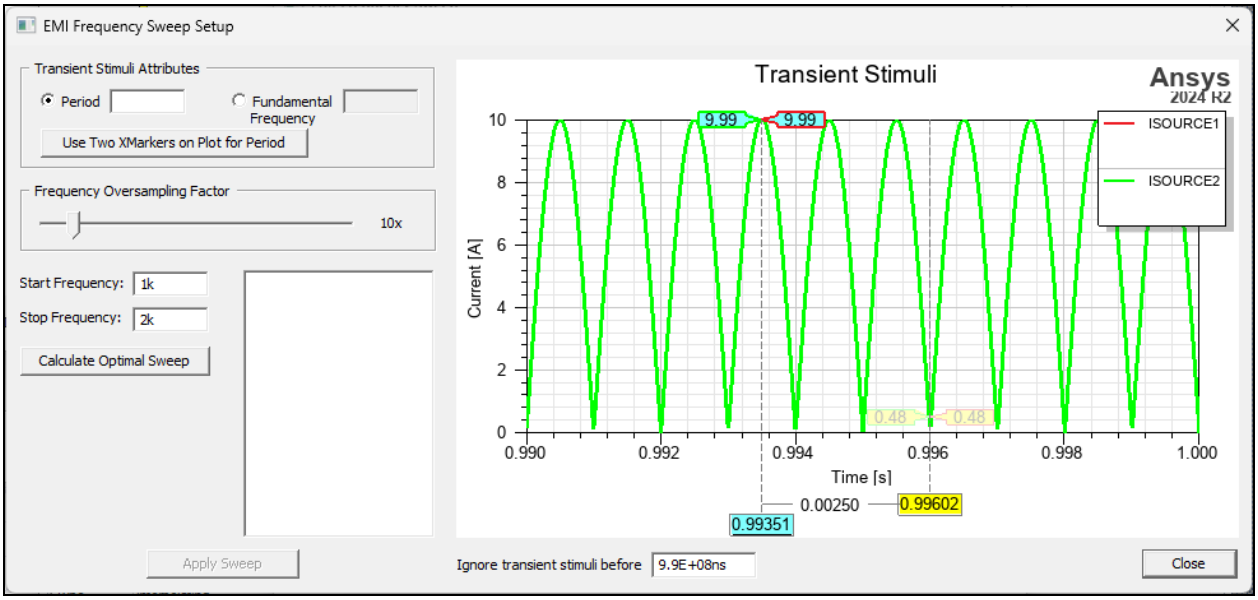


2. Hover directly over the X marker until the cursor becomes a line with two arrows. Then **click+drag** the X marker to the required frequency point.

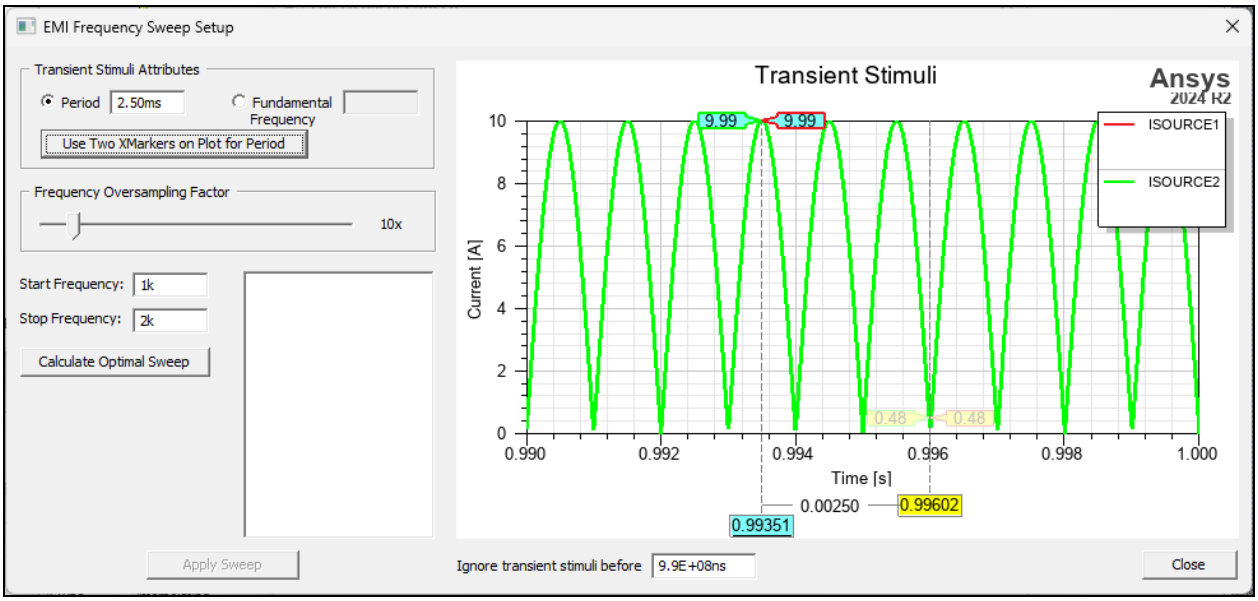


3. Right-click within the graph and select **Add X Marker** again to add a second X marker on the graphic.
4. **Click+drag** the second X marker to the required frequency point.

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 **Transient Stimuli Attributes** area, click **Use Two XMarkers on Plot for Period**. The **Period** box populates with the length between the user-selected frequency points (e.g., **2.50ms**).



Computing DC IR Simulations in HFSS 3D Layout

DC IR analysis solves for the voltage drop across the power plane. The DC solution provides important information pertaining to power delivery, including:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Status of the voltage regulator to determine if it is functioning and supplying the proper DC voltage to all active components attached to power rails
- DC current density information to warn where board or via damage may occur

HFSS 3D Layout makes use of the Ansys SIwave™ DC solver that is based on a well-known formulation and produces an efficient DC solution.

> **Note:**
>
> Thermal modifiers are expressions that evaluate the conductivity of metals as a function of temperature. If such expressions have been specified in the material manager for metals that are used in the design, they are respected during Ansys SIwave™ DC simulations. If no such expressions have been explicitly specified, the Ansys SIwave DC solver uses the following to derate the resistivity of metals as a function of temperature:
>
> resistivity = rho_0 * (1 + 0.004 * (Temp - T0))
>
> where: rho_0 is the metal's resistivity measured at a temperature of T0 = 20 deg C.

HFSS 3D Layout can perform the following actions:

- Add current sources, voltage sources, voltage probes, and DC terminals

  > **Note:**
  >
  > DC terminals are not currently supported within the IC Mode environment. The terminal icon will not appear in either the menu or ribbon.

- Add a transient excitation
- (**General Mode Only**) View and create Voltage Regulators (VRMs)
- (**General Mode Only**) Define equipotential regions
- Compute DC current and voltage distribution
- Export DC IR simulation results
- (**General Mode Only**) Export DC IR data to Icepak, or import a temperature map from Icepak to use in the DC IR simulation
- Plot DC IR field overlays
- (**General Mode Only**) Overlay a DC Temperature Field

From General mode, users can configure component parameters prior to simulation from the **SIwave Workflow Wizard** or skip directly to Setting Up the Compute DC Current and Voltage Distribution Window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Computing DC IR Simulations in IC Mode

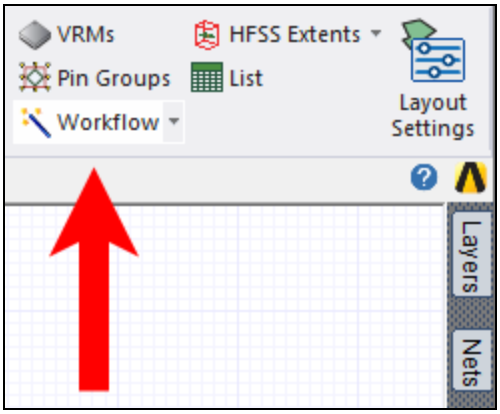
Users can create DC IR analysis solves in both General mode and IC mode. IC Mode integrates ECAD Xplorer into the HFSS 3D Layout environment, preserving ECAD Xplorer's capabilities of creating larger, more complex IC models, while allowing users direct access to most General Mode features (Refer to Layout Editor), with some notable differences:

- The SIwave Workflow Wizard is **not** available in IC Mode. IC Mode users can skip directly to Setting Up the Compute DC Current and Voltage Distribution Window.
- Voltage sources, current sources, and voltage probes are supported. DC terminals are **not** currently supported.
- The **Compute DC Current and Voltage Distribution** window contains a subset of options available in General mode. Options unique to General mode are noted in the instructions.
- All simulation results are available to view and export, except the DC Power Tree.
- Overlapping stackups **are** supported. In General Mode, only laminate stackups are supported.

Beginning Simulation From the SIwave Workflow Wizard (General Mode Only)

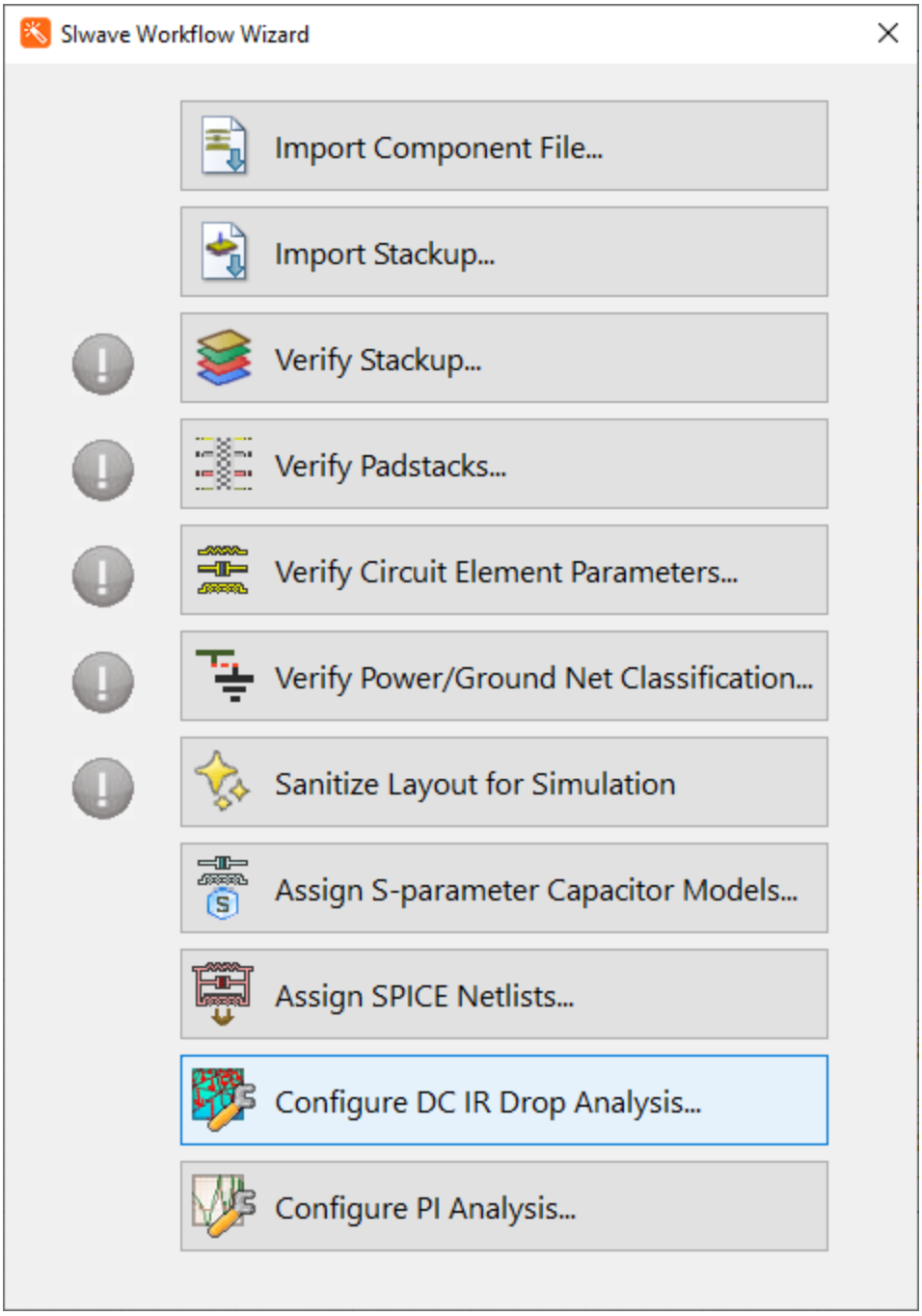
Follow these steps to begin configuring a simulation from the **SIwave Workflow Wizard**.

1. From the **Layout** tab, select **Worfklow** to open the **SIwave Workflow Wizard** window.



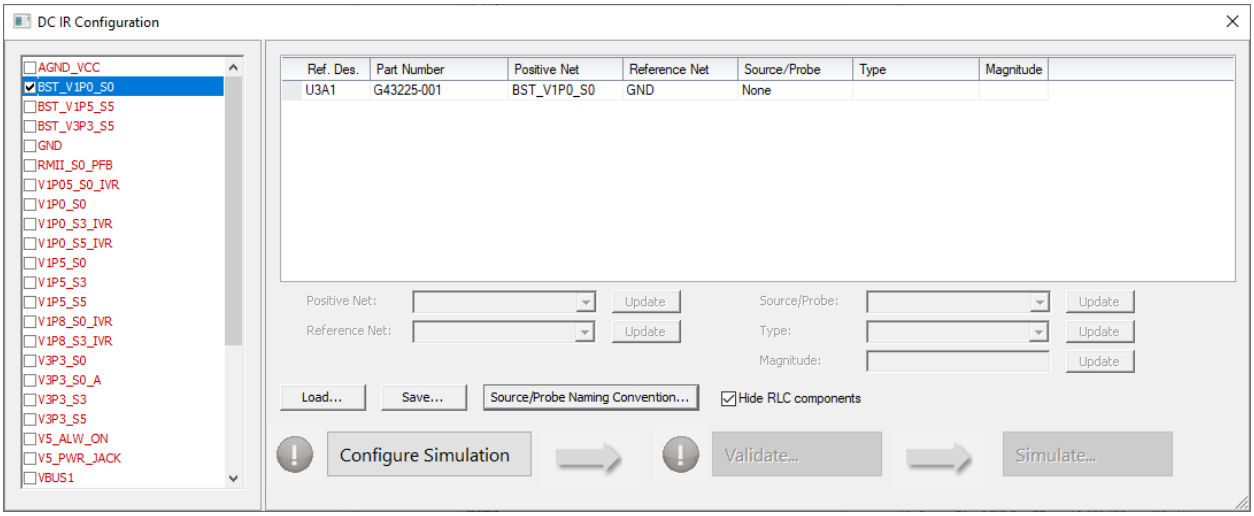
2. Click **Configure DC IR Drop Analysis** to open the **DC IR Configuration** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

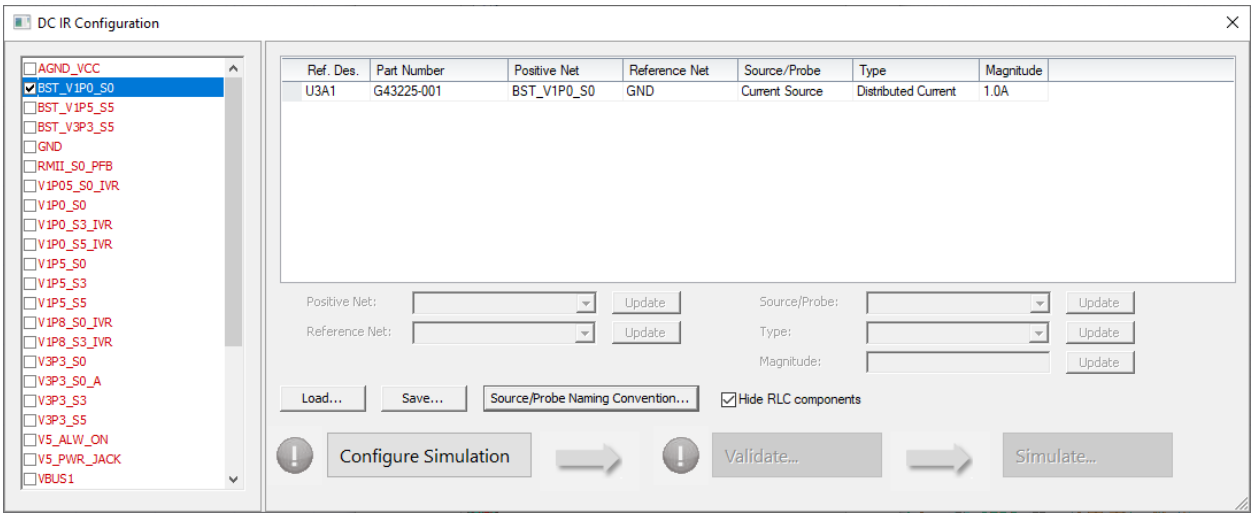


3. From the list on the left, check the box adjacent to a selected net (e.g., **BST_V1P0_S0**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



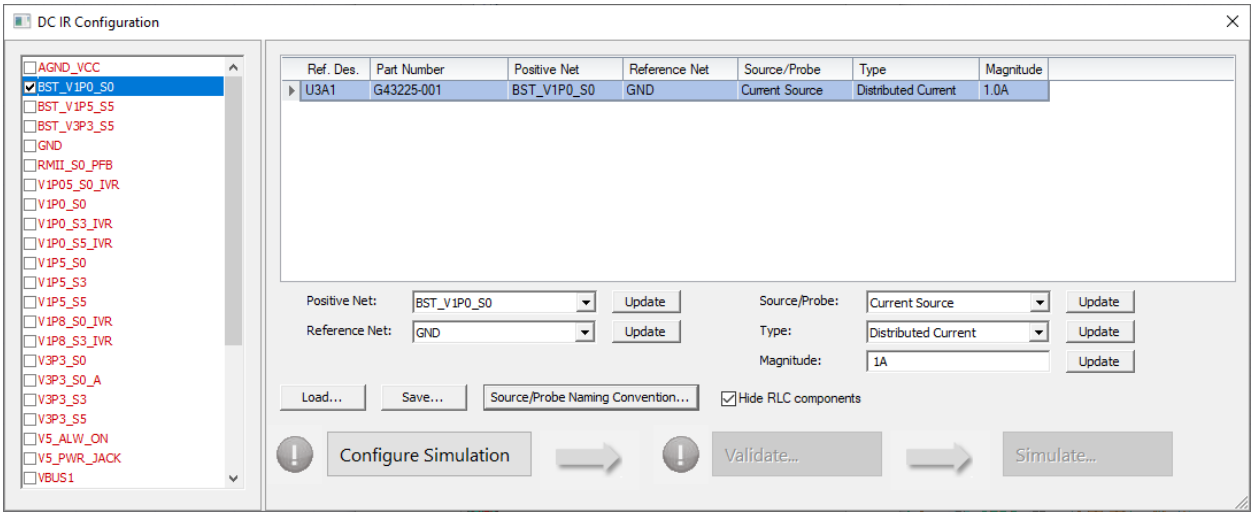
4. From the table, do the following:
   a. Make selections from the **Source/Probe** drop-down menu (i.e, select from **None**, **Current Source**, **Voltage Source**, **Voltage Probe**, or **Terminal**).
   b. The **Type** drop-down menu is enabled if **Current Source** or **Voltage Probe** are selected from the **Source/Probe** drop-down menu. Select either **Constant Voltage** (i.e., adding one circuit element) or **Distributed Current** (i.e., adding a circuit element on every pin) from the **Type** drop-down menu.



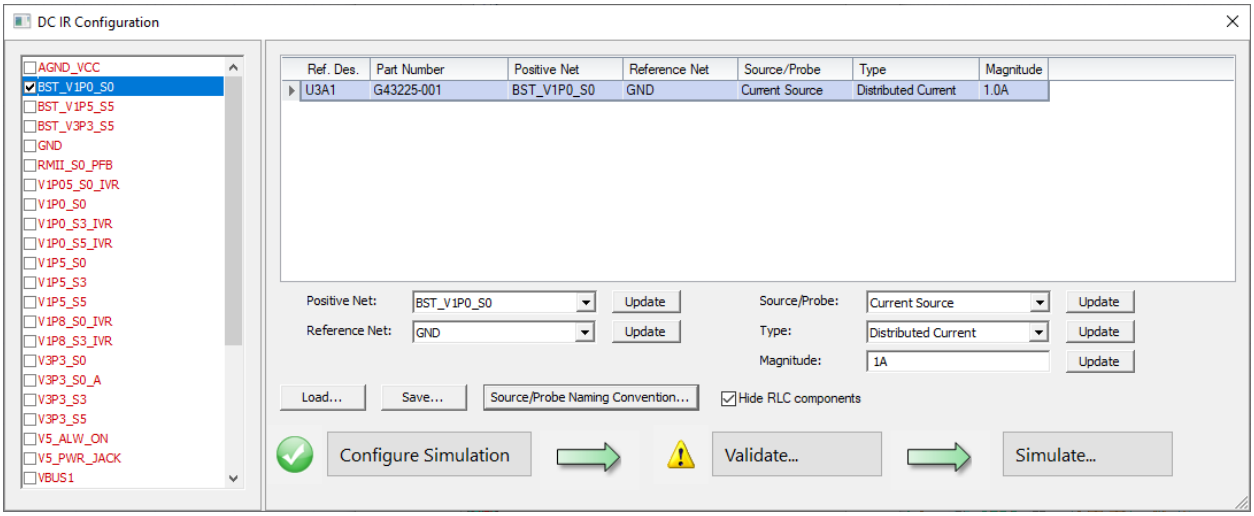
   c. Enter a **Magnitude** in the adjacent field or accept the default parameter, if any.
   d. Select the left-most empty cell in the component's row to populate the fields

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

beneath the table. Then make further adjustments, as appropriate.

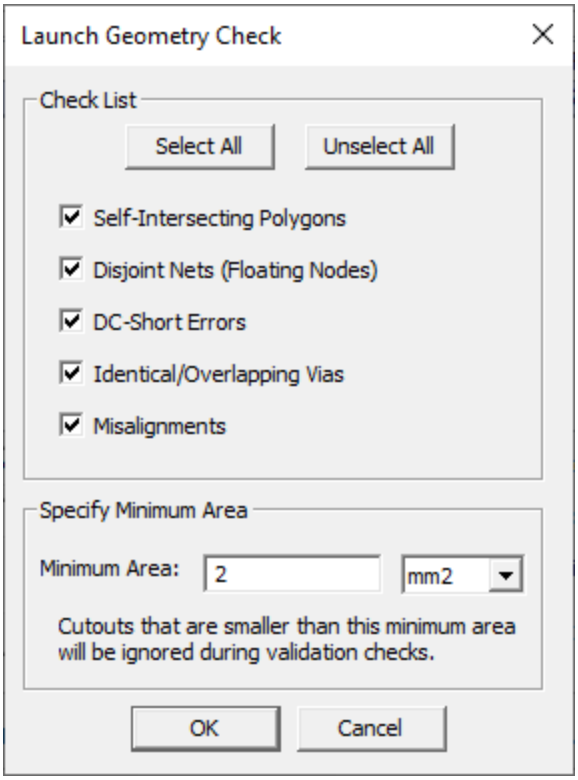


5. Click **Configure Simulation**.

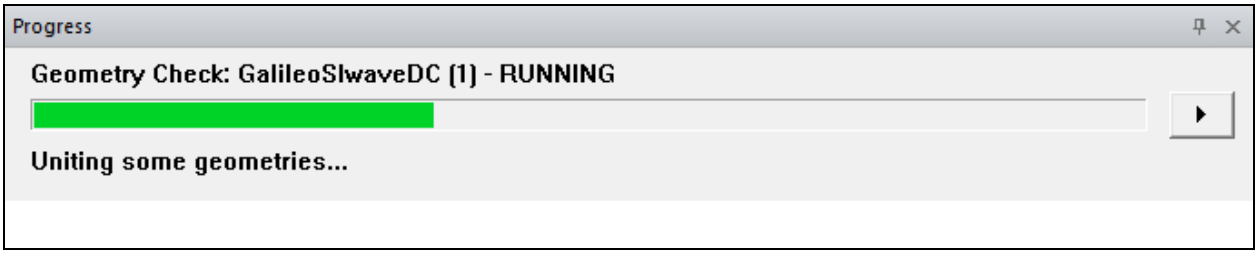


6. Click **Validate** to open the **Launch Geometry Check** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



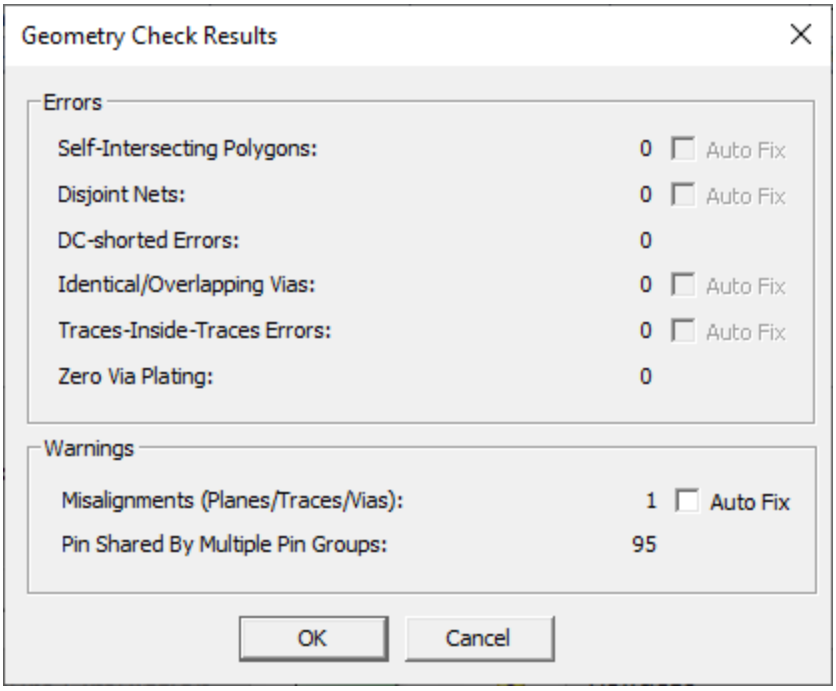
7. If applicable, configure the validation check. Then click **OK** to begin validation. View progress in the **Progress** window.



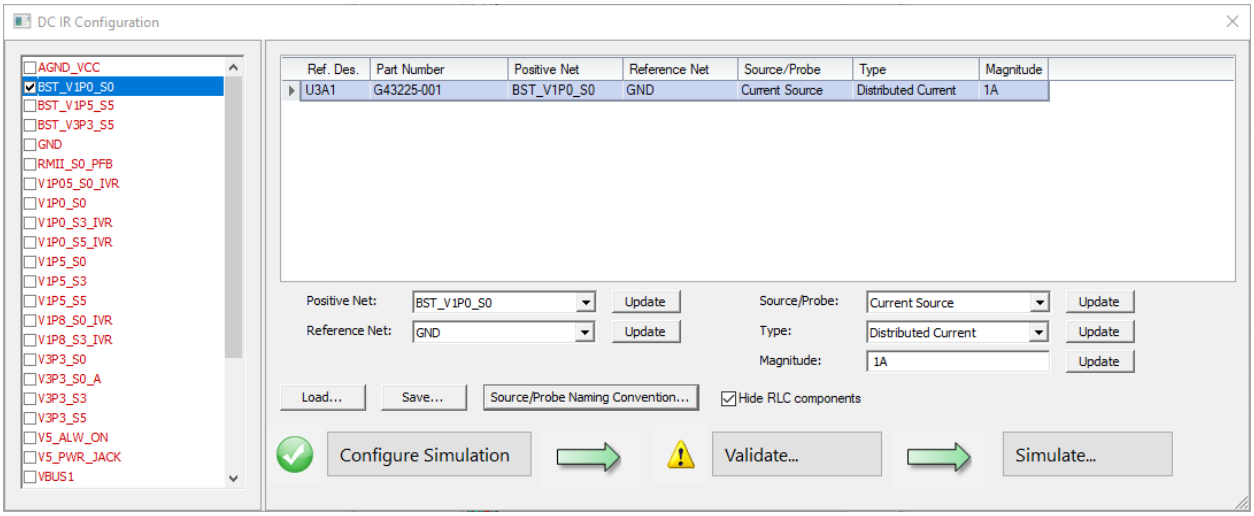
> **Note:**
>
> Users will need to open the **Progress** window and position it conveniently beforehand.

8. After the validation check is complete, a **Geometry Check Results** window appears. Assuming the simulation has passed validation, click **OK** to return to the **DC IR Configuration** window.

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 **Simulate** to open the **Compute DC Current and Voltage Distribution** window.



10. Continue to step 2 of "Computing DC IR Simulations in HFSS 3D Layout " on page 10-243.

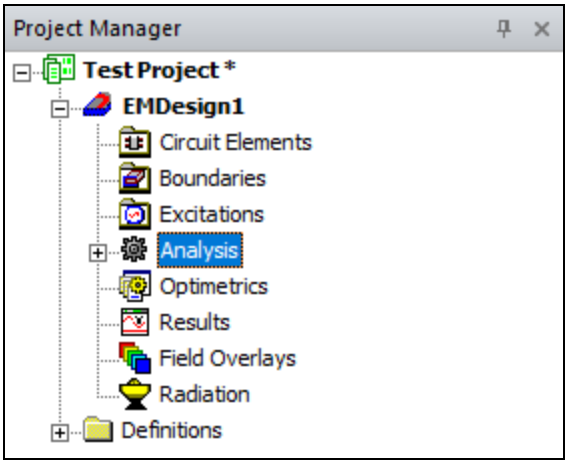
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up the Compute DC Current and Voltage Distribution Window

Follow these steps to finalize simulation solver setup and run a simulation computing DC current and voltage distribution.

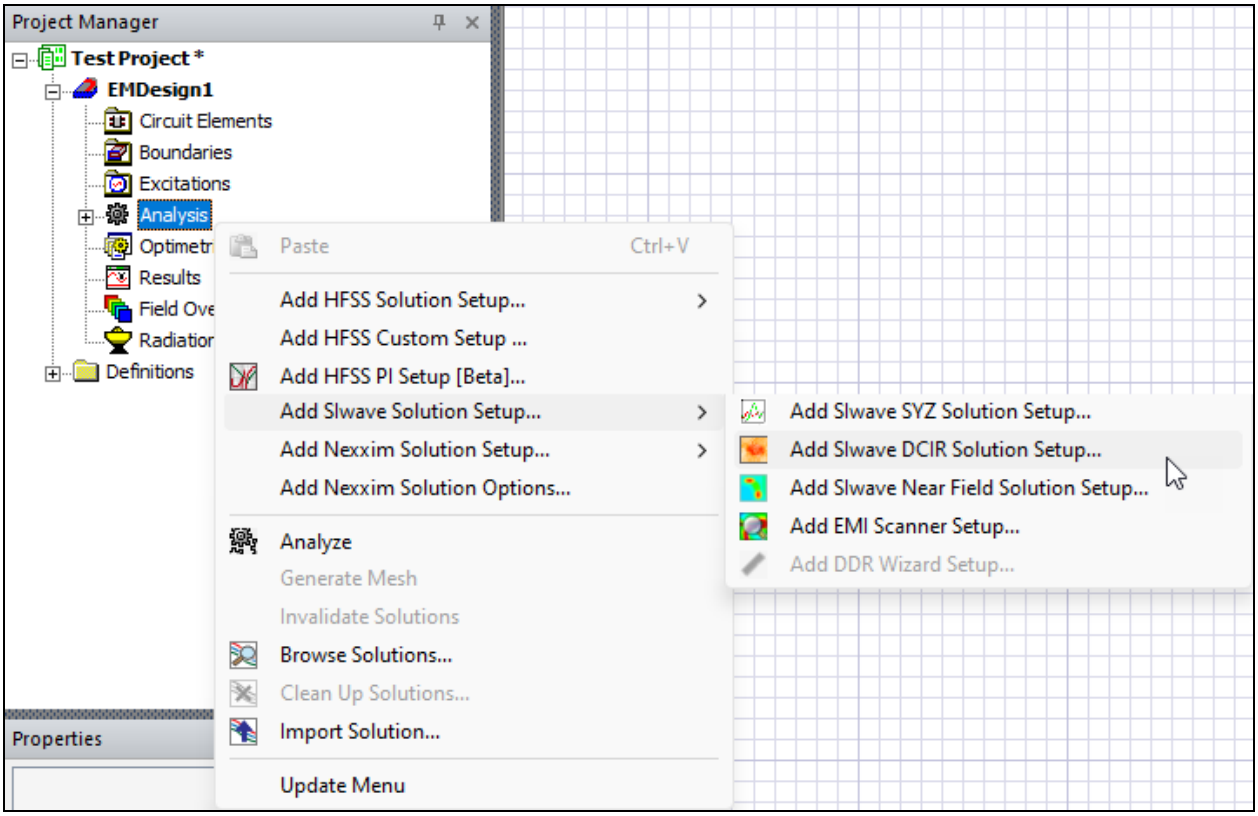
> **Note:**
>
> Some options are only available from the General mode or IC mode environments and will be absent from the UI.

1. From the **Project Manager** window, expand the **Project Tree** and ***Active Design Folder***. Then right-click **Analysis**.



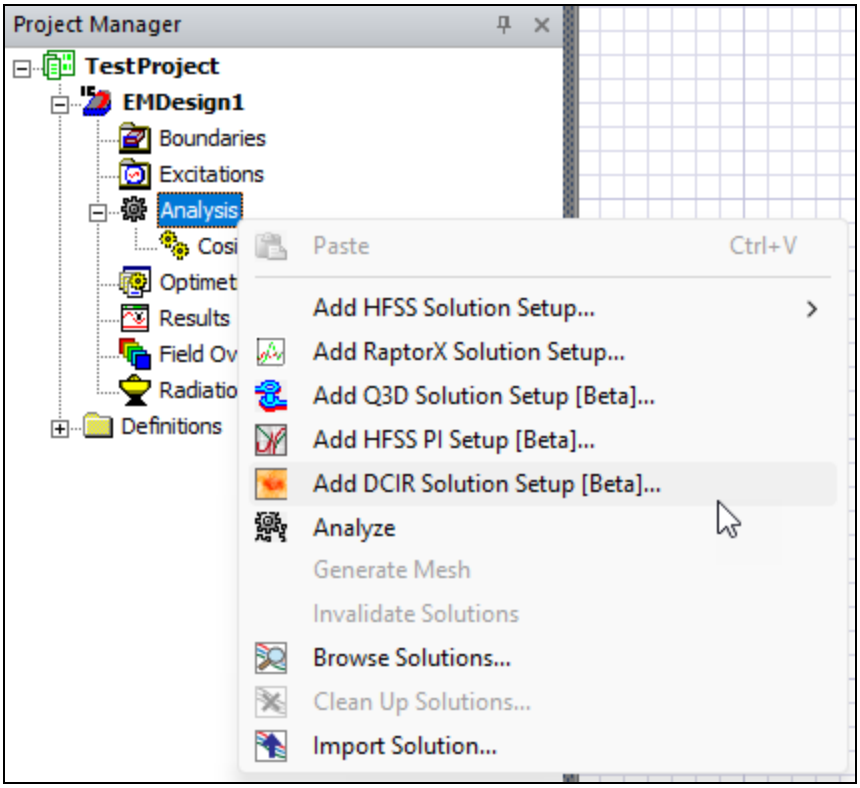
2. Do one of the following to open the **Compute DC Current and Voltage Distribution** window.
   - From General mode, select **Add SIwave Solution Setup** > **Add SIwave DCIR Solution Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

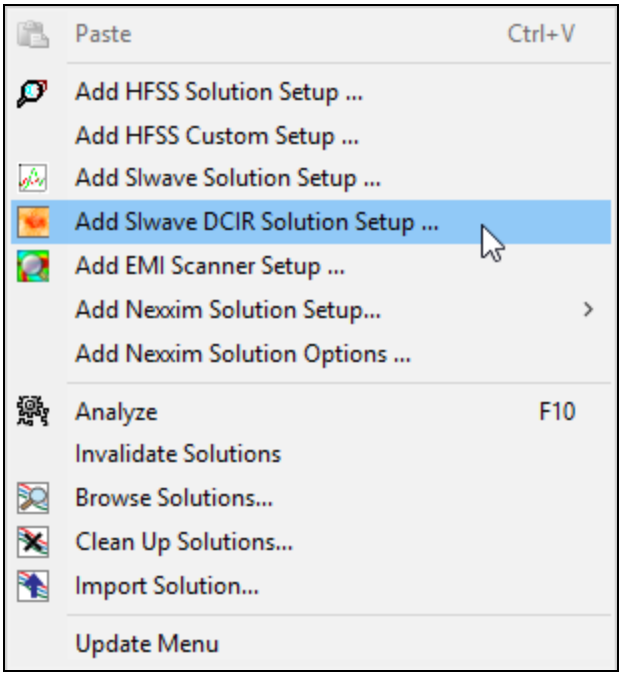


- From IC Mode, select **Add DCIR Solution Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

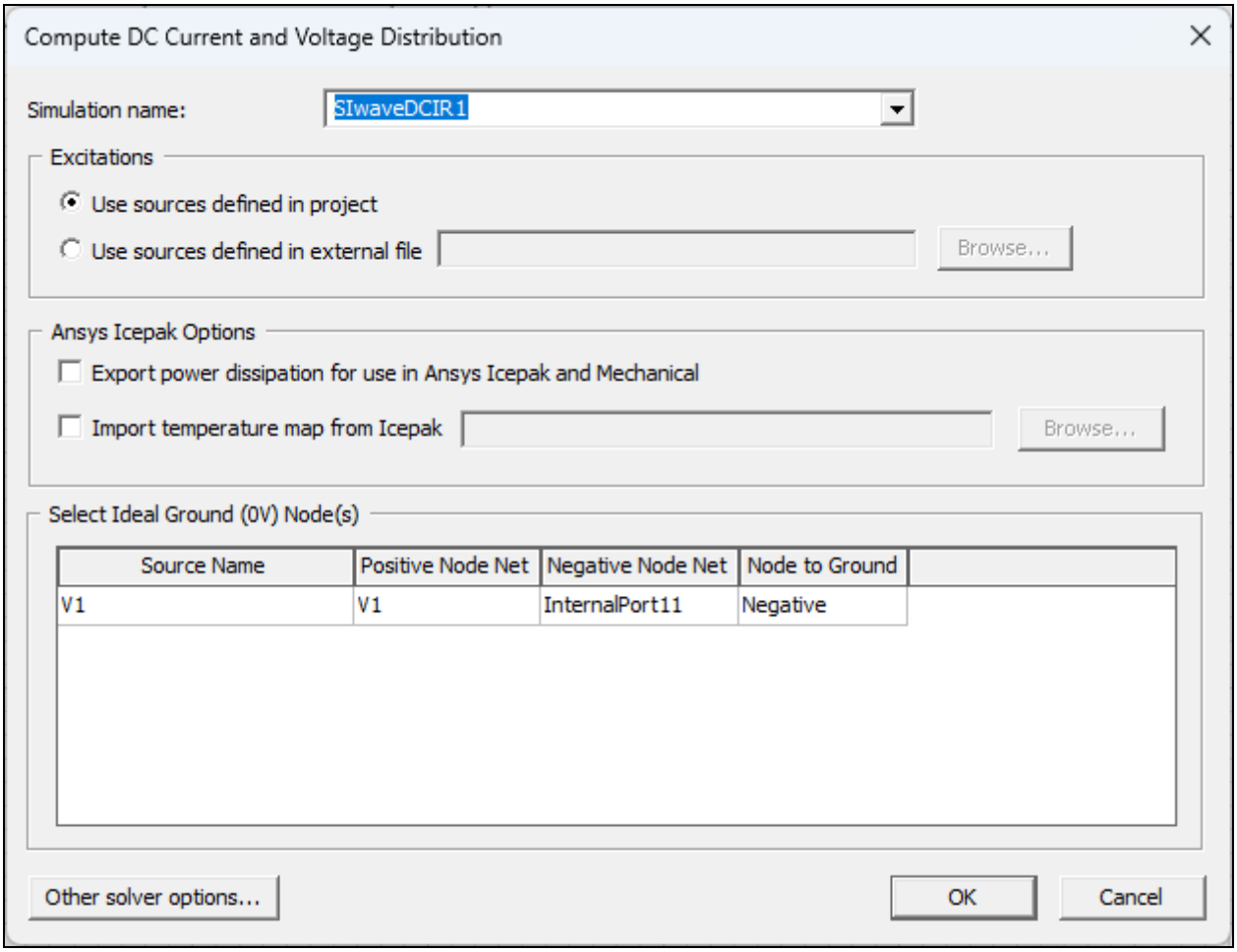


Then select **Add Slwave DCIR Solution Setup** to open the **Compute DC Current and Voltage Distribution** window.



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 **Compute DC Current and Voltage Distribution** window, do the following:



- **Simulation name** – enter a name for the DC IR simulation.
- **(General Mode Only)** From the **Excitations** area, select a source of excitations (i.e., **Use sources defined in the project** or **Use sources defined in external file**). If the user selects **Use source defined in external file**, **Browse** to an appropriate file and select it (e.g., **test.tmp**).
- **(General Mode Only)** From the **Ansys Icepak Options** area, there are two optional settings:
  - Check **Export power dissipation for use in Ansys Icepak® and IcepakFEA** to export data to either of these design types.
  - Check **Import temperature map** to select an Ansys Icepak/Ansys SIwave temperature distribution file, then **Browse** to an appropriate file and select it (*test.sitemp). The temperature data can locally adjust DC conductivity.
- From the **Select Ideal Ground Node(s)** area, select a ground node on the table. Select a current or voltage source at 0 volts (ideal ground). All other voltages are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

computed with respect to this node. If no ideal ground node is selected, HFSS 3D Layout automatically picks a source terminal to ground.

Click the **Node to Ground** drop-down menu to select **Neither**, **Negative**, or **Positive**.

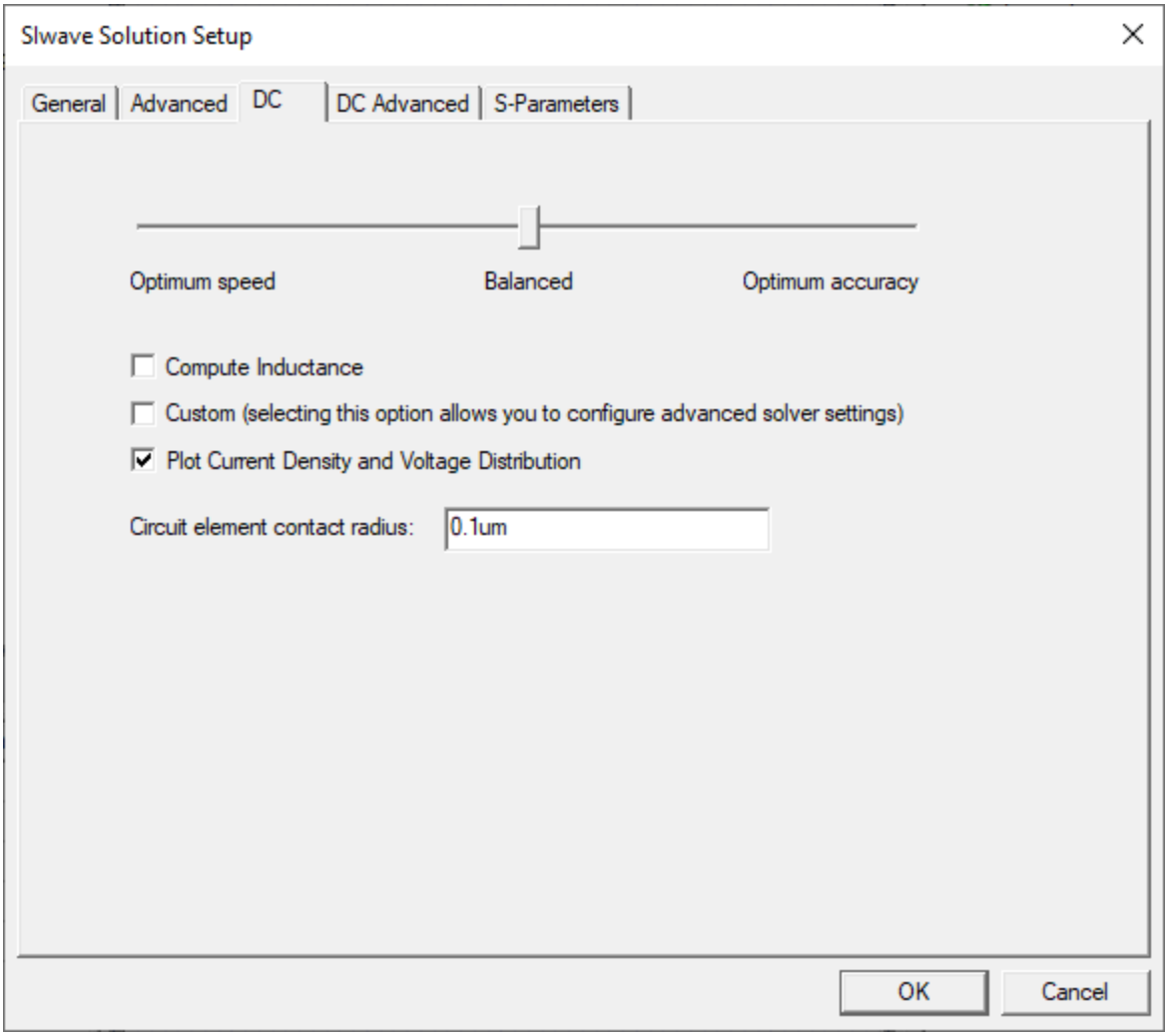
If a **Negative** voltage source is selected, the voltage display on its ground pin is 0 V.

If a **Positive** voltage source is selected, the voltage display on its ground pin are negative.

> **Note:**
>
> Assume that the selection is the 0 node in a SPICE simulation. This is the node to which all voltages should be referenced. Since HFSS 3D Layout includes the return paths of all planes (including ground), do not assume all the sources reference that 0 node. An absolute 0 node reference must be selected. So if the 0 node is on the positive side of a voltage plane, the "ground" or return path is shown as negative.

4. To view additional solver options, click **Other solver options**. The **SIwave Solution Setup** window opens, on the **DC** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



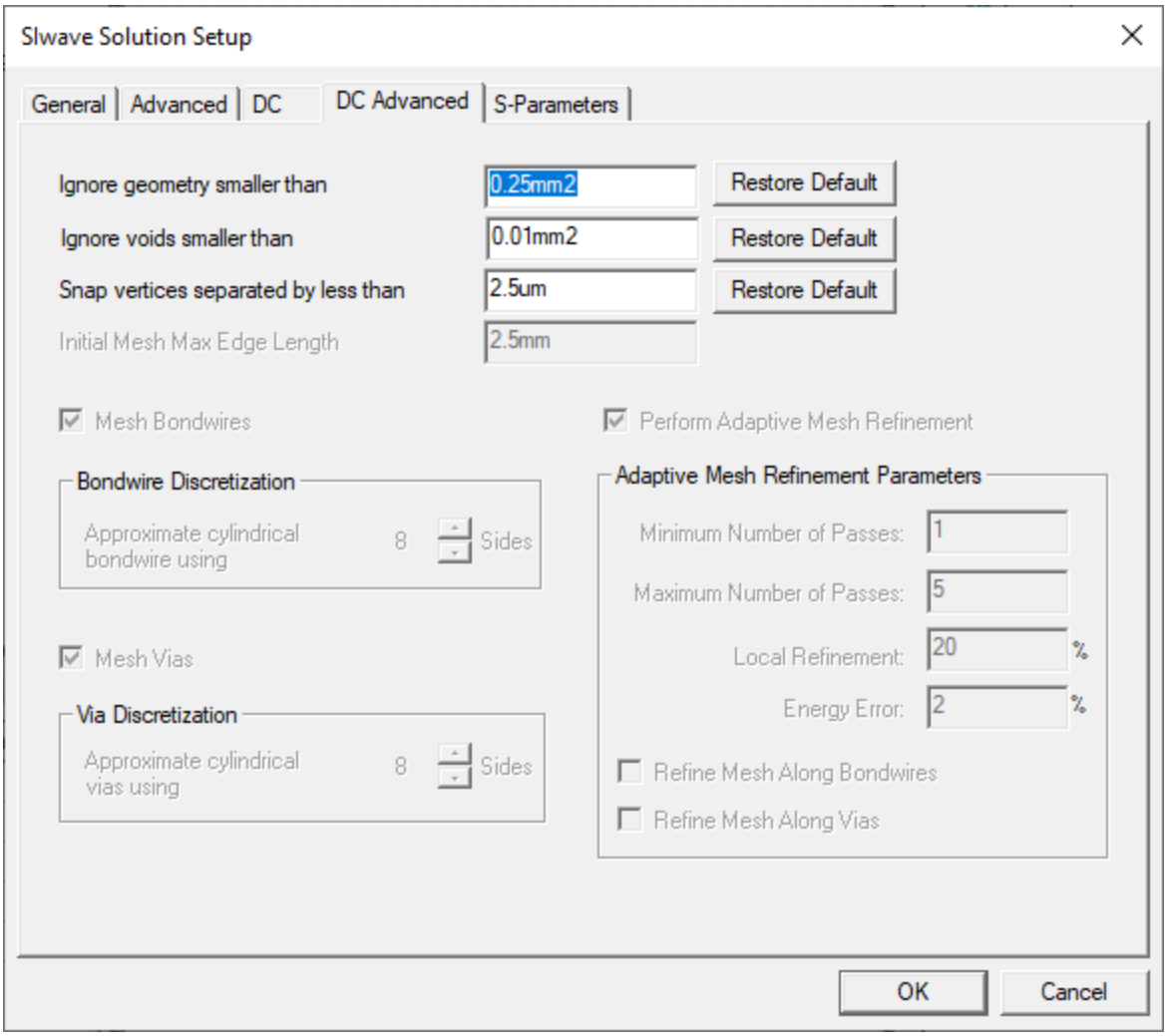
From this window, do the following:

- Use the slider to balance speed and accuracy:
  - **Optimum Speed** – all options in the **DC Advanced** tab are deactivated.
  - **Balanced** – the Mesh Bondwires and Mesh Vias options are selected, along with adaptive Mesh Refinement.
  - **Optimum Accuracy** – the mesh are refined on bondwire and vias, and more aggressive refinement parameters are specified.
- Select whether to **Compute Inductance**. The inductance is computed by integrating the vector potential at DC using an efficient, multi-level fast multipole method with two-level parallelization. In some cases, this may require significant computational time and memory.
- Select whether to use **Custom** simulation settings.

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 **Plot Current Density and Voltage Distribution**. Clear this check box to save disk space if users do not need to post-process voltage and current plots (e.g., if the user only want to export a SPICE netlist after a DC simulation, it might not be helpful to save any field data to disk).
- Enter the **Circuit Element Contact Radius**. Equal potential is enforced in the contact radius, and no mesh refinement is done inside the contact region.

5. Check **Custom** to enable DC Advanced settings, then select the **DC Advanced** tab.



DC Advanced options include:

- **Ignore geometry smaller than** – geometry whose area (after being combined with any overlapping geometry) falls under this specified threshold is ignored during simulation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

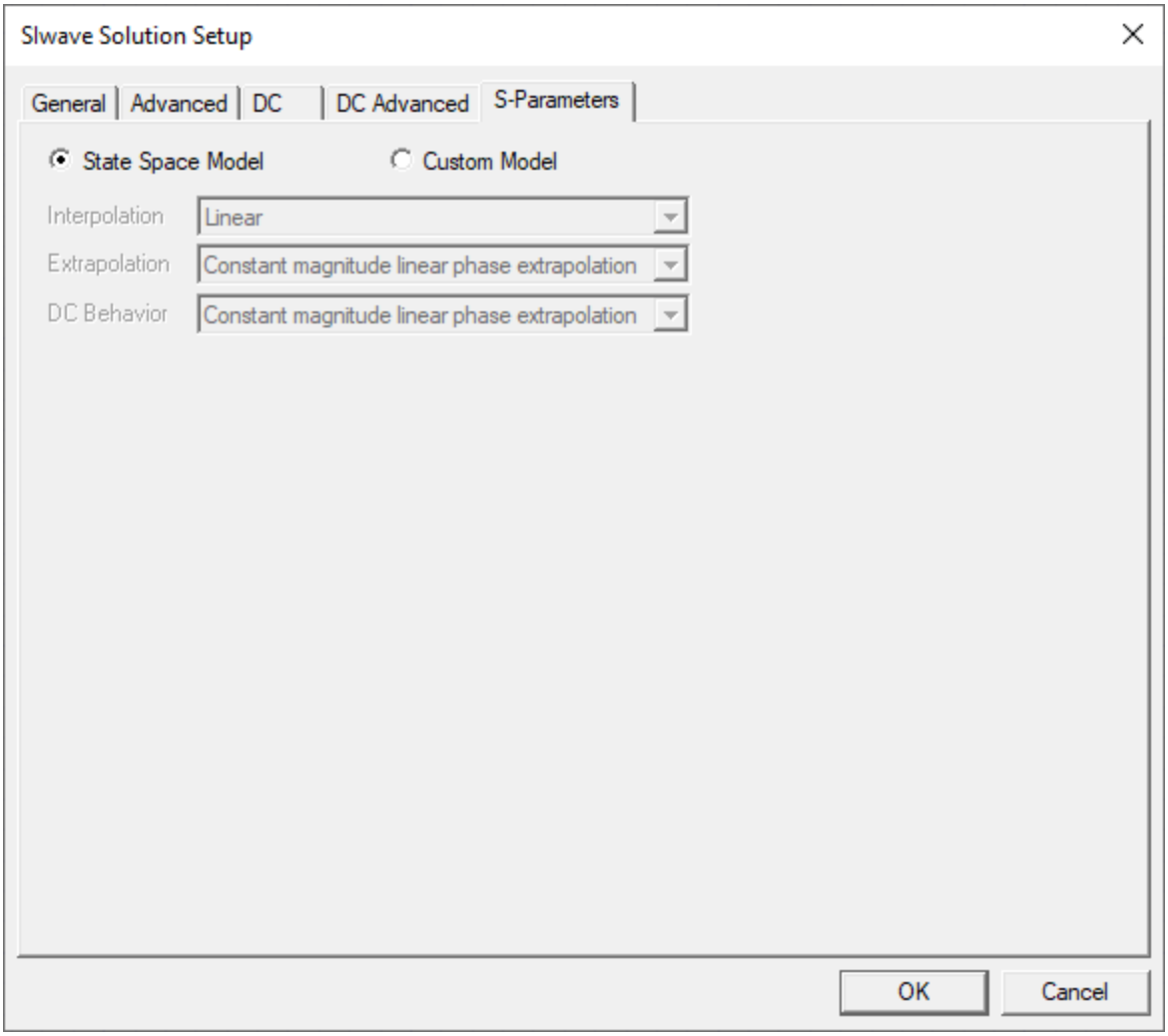
- **Ignore voids smaller than** – geometry whose area (after being combined with any overlapping geometry) falls under this specified threshold is ignored during simulation.
- **Snap vertices separated by less than** – entities (e.g., vias) and vertices that are separated by less than the distance specified in this box may be snapped together during solver pre-processing.
- **Initial Mesh Max. Edge Length** – specifies the maximum edge length selected in the mesh (before any adaptive refinement occurs).
- **Mesh Bondwires** – check if users do not want to model bondwires as point connections to mesh. Specify the **Bondwire Discretization** (the number of sides used to approximate cylindrical bondwires). Specify any value between 6 and 64. The default value is 8.
- **Mesh Vias** – check if users do not want to model vias as point connections to mesh. Specify the **Via Discretization** (the number of sides used to approximate cylindrical vias). Specify any value between 6 and 64. The default value is 8.
- **Perform Adaptive Mesh Refinement** – check to perform Adaptive Mesh Refinement, and populate the fields:
    - **Minimum Number of Passes** – an adaptive analysis does not stop unless the minimum number of passes specified has been completed, even if convergence criteria have been met.
    - **Maximum Number of Passes** – this value is the maximum number of mesh refinement cycles users want the simulation to perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops; otherwise, the adaptive analysis continues unless the convergence criteria are reached.

      > **Note:**
      >
      > The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform requesting more memory than is available or taking excessive time to compute solutions.

    - **Local Refinement** – this value determines how many triangles are added at each iteration of the adaptive refinement process. The triangles with the highest error is refined. The default value is 20%.
    - **Energy Error** – the default value is 2%.
- **Refine Mesh Along Bondwires** – check to refine mesh along bondwires.
- **Refine Mesh Along Vias** – check to refine mesh along vias.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Select the **S-Parameters** tab.

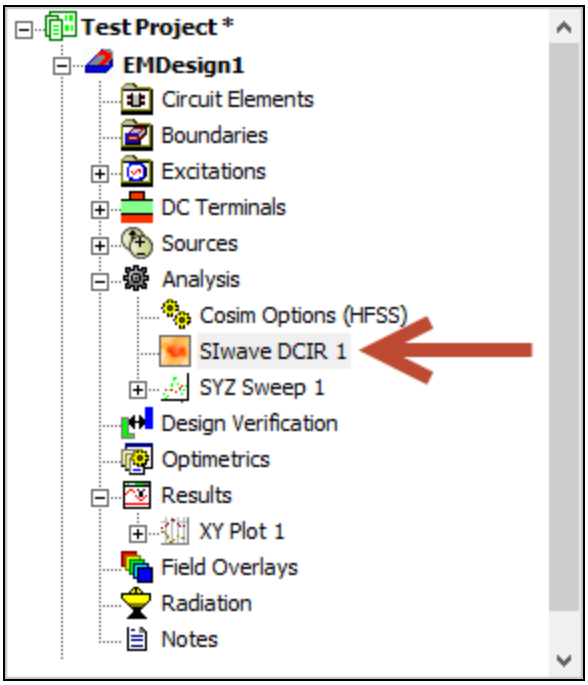


7. Select **State Space Model** or **Custom Model**. If **Custom Model** is selected, set the following options:
   - **Interpolation** – select **Point**, **Linear**, or **Step** from the drop-down menu.
   - **Extrapolation** – select **Zero padding**, **Same as last point**, **Linear extrapolation from last 2 points**, or **Constant magnitude linear phase extrapolation** from the drop-down menu.
   - **DC Behavior** – select **Zero padding**, **Same as last point**, **Linear extrapolation from last 2 points**, or **Constant magnitude linear phase extrapolation**, **Series Capacitor**, or **Leave all signal lines open circuited** 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.

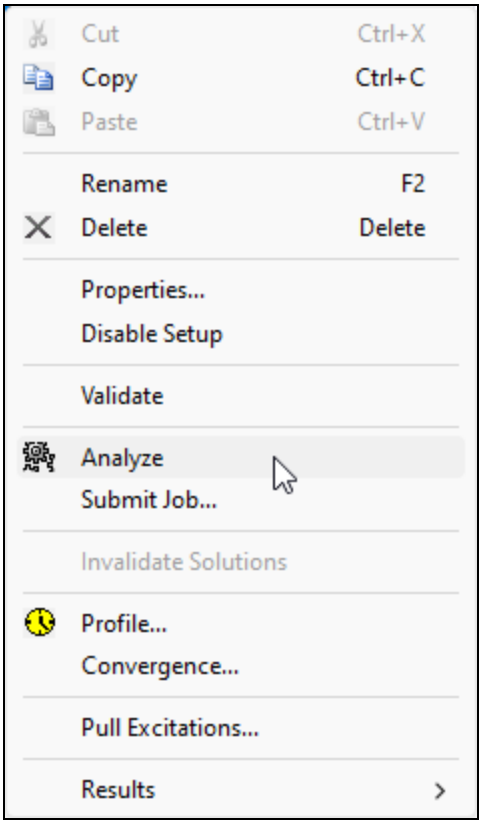
> **Note:**
>
> Refer to the following pages for an example of using the **SIwave Solution Setup** window to change S-Parameters or using a preconfigured Python script to generate an Ansys ECAD database file, which can then be imported into AEDT to create a custom Ansys SIwave setup.

8. After making any selected changes, click **OK** to exit the solution setup.
9. Click **OK** to save the solution setup. The **Project Manager** window updates to show the DC IR setup under **Analysis**.



10. From the **Project Manager** window, right-click the analysis and select **Analyze**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



11. The **Progress** window displays the simulation's progress. When it finishes, view or export results.

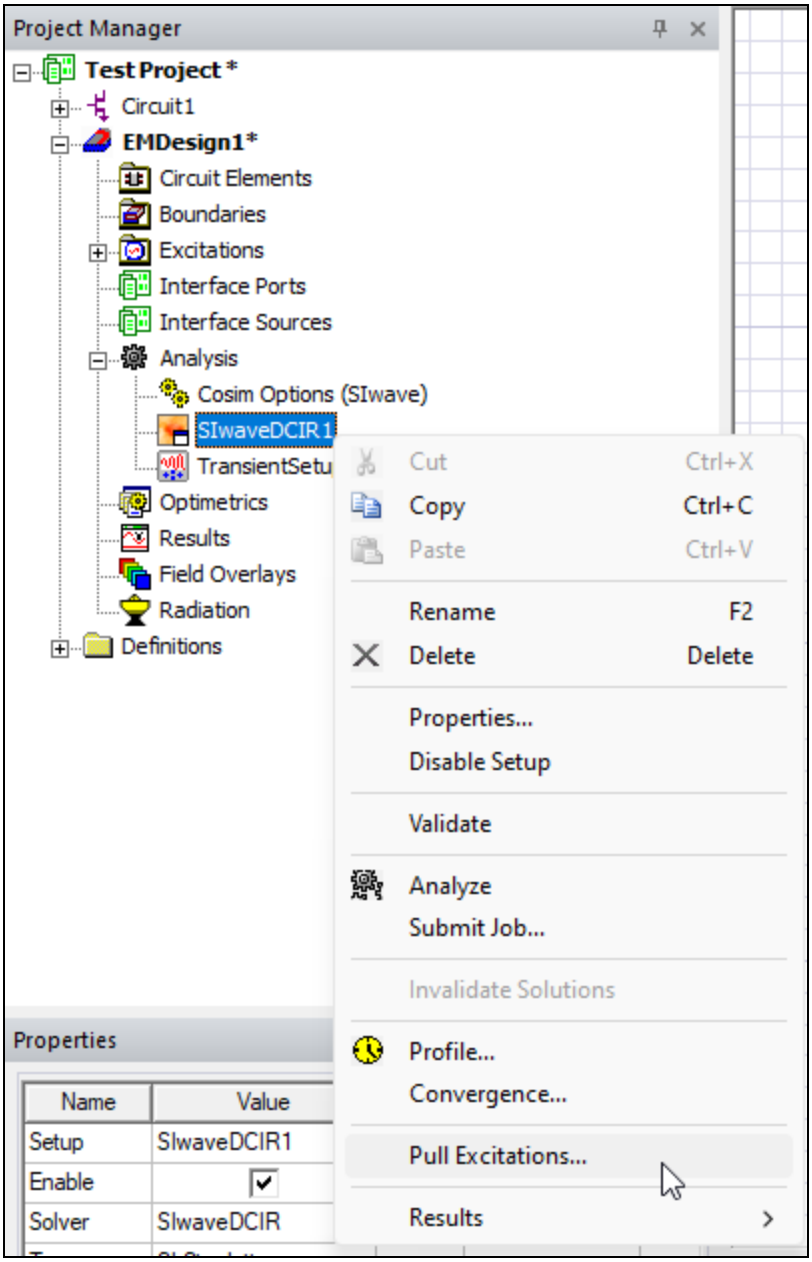
Assigning a Transient Excitation

These instructions assume the user has already set up and solve a Transient Analysis on the design with all appropriate drivers and receivers attached to ports. Transient currents flowing into each port are captured and converted to frequency-domain sources during execution of DCIR simulations. Users can also record these steps using the **Record Script** function on the **Automation** tab, then run the resulting Python script to set the specified excitation source. Refer to Using a Python Script to Configure an SIwave Solution Setup.

Follow these steps to select a Circuit or HFSS 3D Layout transient excitation source to use as an external source for simulation.

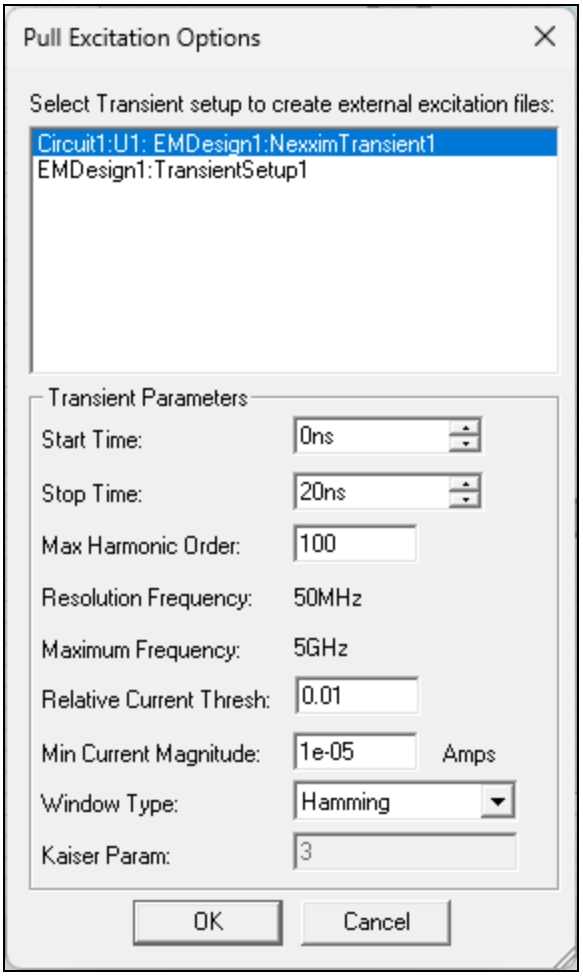
1. From the **Project Manager** window, expand the active design folder (e.g., **Test_Project**) > subdesign (e.g., **EMDesign1**) > **Analysis**. Then right-click the DCIR solution setup (e.g., **SIwaveDCIR 1**) and select **Pull Excitations** to open the **Pull Excitation Options** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Pull Excitation Options** window, do 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.



a. Select a Transient setup to create external excitation files (e.g., **Circuit1:U1:EMDesign1:NexximTransient1**).

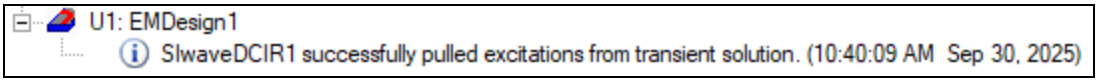
b. Make any appropriate changes in the **Transient Parameters** area. By default, the data that is pushed to a dynamically linked project is automatically calculated. The start and stop times must lie within the transient simulation time limits.

- **Start Time** – enter the beginning time of the transient interval.
- **Stop Time** – enter the ending time of the transient interval.
- **Max Harmonic Order** – specify the maximum number of harmonics to include when calculating frequency-domain data. By default, this value is set to half of the transient simulation's sample points, corresponding to the Nyquist frequency. Together with **Start** and **Stop Time**, this determines **Resolution Bandwidth** and **Maximum Frequency**.

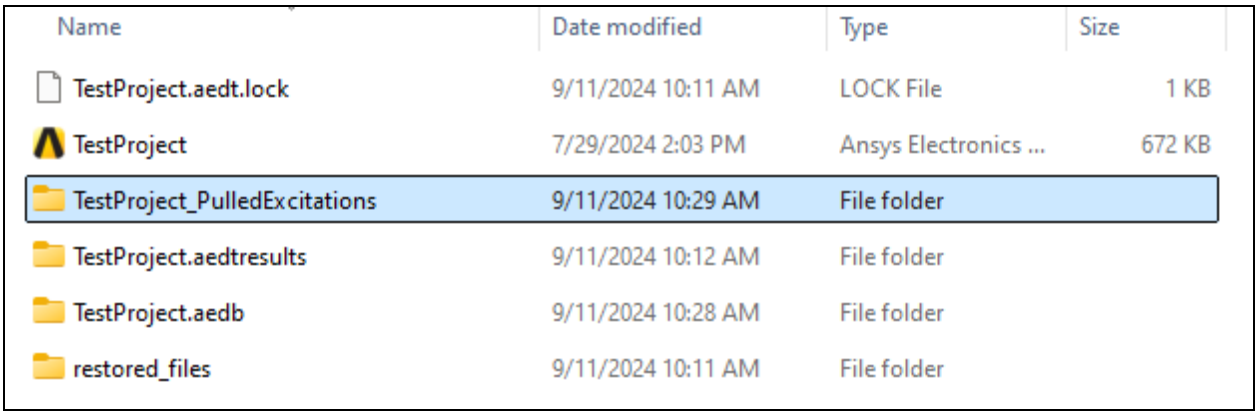
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Reference Frequency** – displays the frequency used as the reference point for harmonic calculations.
- **Maximum Frequency** – displays the highest frequency included in the data.
- **Relative Current Threshold** – enter the minimum relative current level to include in the excitation data. Signals below this threshold are ignored.
- **Min Current Magnitude** – specify the minimum current magnitude to include in the excitation data.
- **Window Type** – select a windowing function from the drop-down menu (e.g., Hamming, Hanning, Kaiser, Welch, etc.). Non-rectangular windows apply a processing loss (coherent gain), which is automatically compensated when excitations are pushed to linked projects.
- **Kaiser Param** – if **Kaiser** is selected from the **Window Type** drop-down menu, enter the beta parameter that controls the trade-off between main-lobe width and side-lobe level. Only editable if **Kaiser** is selected.

3. Click **OK** to close the **Pull Excitation Options** window. If the excitation is accepted, a confirmation message appears in the **Message Manager** window.

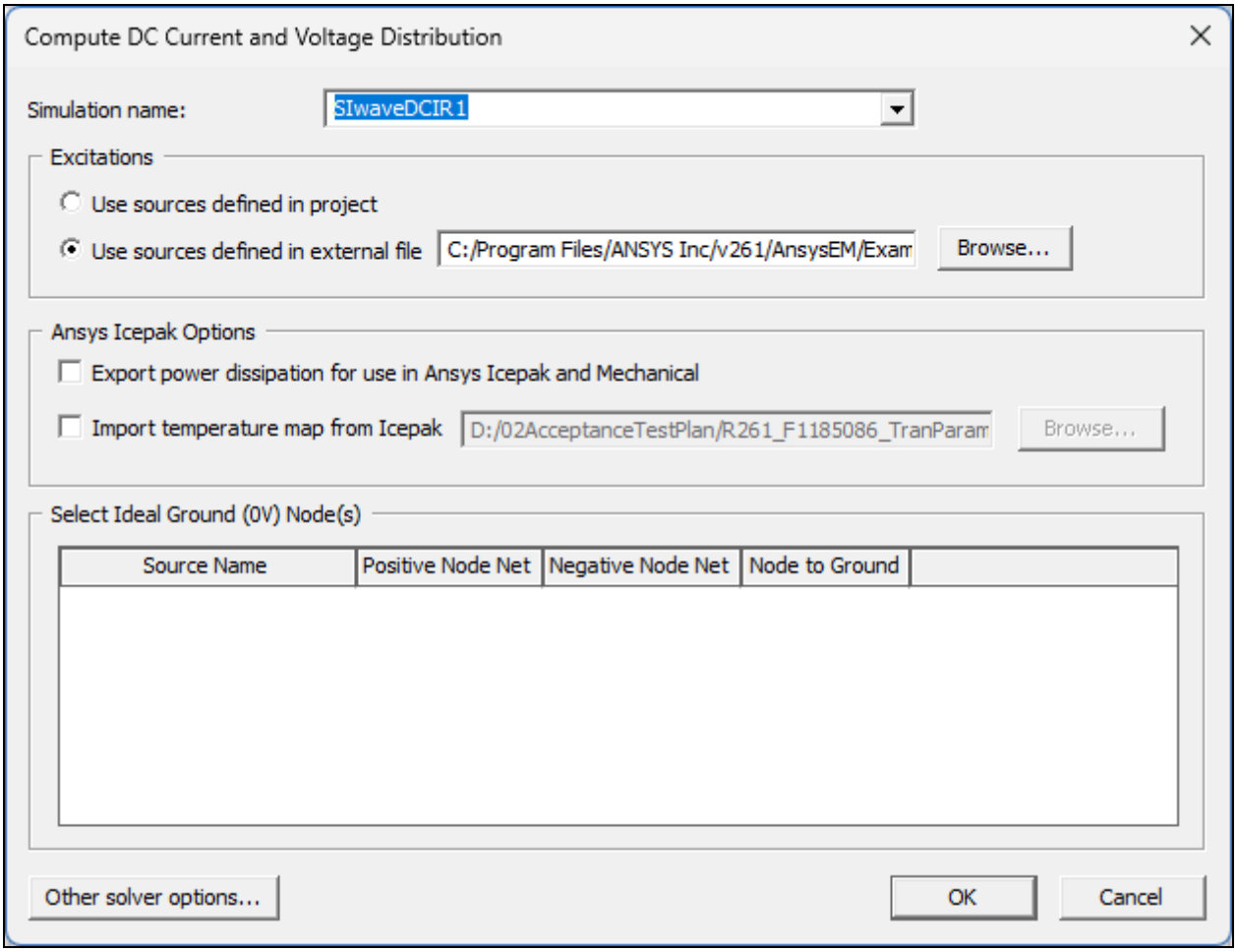


Within the same directory where the project is saved, a folder is created called ***<ProjectName>*_PulledExcitations**, which contains the newly created excitation files referred to in the **Message Manager** window.



From the **Compute DC Current and Voltage Distribution** window, **Use sources defined in external file** is now selected and the adjacent field is populated with the path to the excitations source file.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Adding Sources, Probes, and Terminals

The DC IR workflow wizard's **Configure DC IR Analysis** window can be used to generate any of the new element types attached to pin groups of existing components. Alternatively, the new elements can be generated from the **Layout** ribbon.

Both source types (voltage or current) and voltage probes are modeled as two terminal elements, similar to circuit ports, and are generated in a similar way. Sources can be connected to padstack instances, pin groups, user-specified points in the model space, or components. Refer to Creating Sources on Components.

Adding Voltage Sources, Current Sources, and Voltage Probes

The following example explains how to add a voltage source to the project. Complete the same steps to add as many voltage sources, current sources, voltage probes, or DC terminals as

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

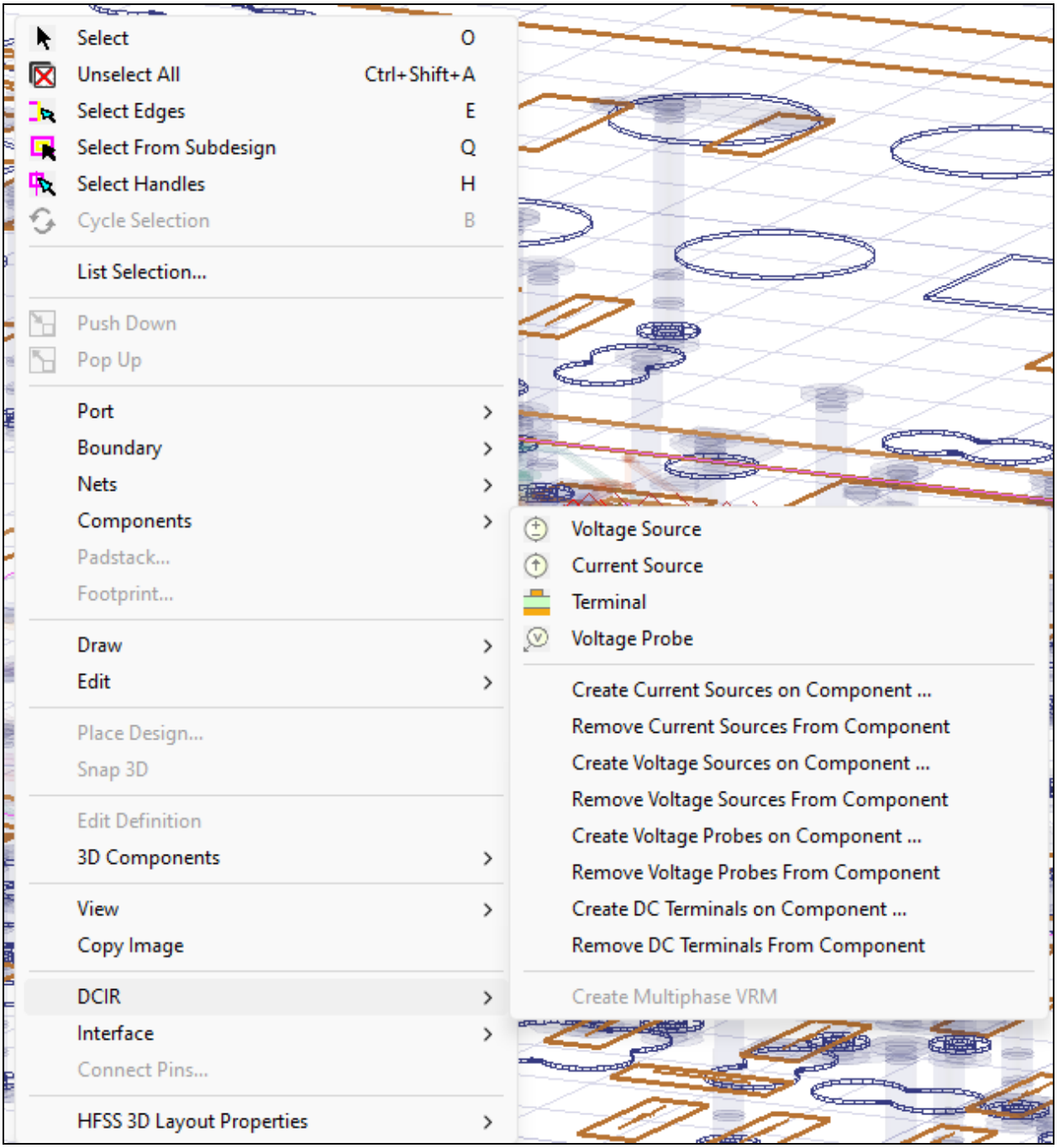
necessary.

> **Note:**
>
> DC terminals are not currently supported within the IC Mode environment. The terminal icon will not appear in either the menu or ribbon.

1. From the **Layout Editor**, do one of the following to add a voltage source, current source, voltage probe, or DC terminal.

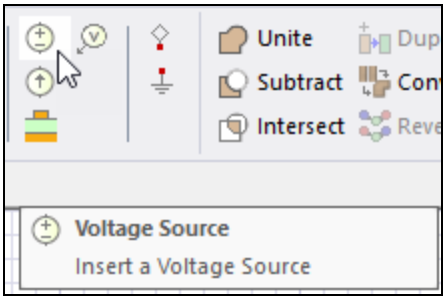
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From within the design area, right-click and select the appropriate element.



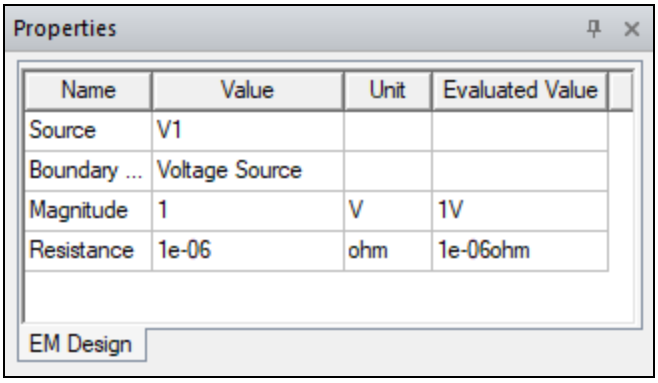
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the **Layout** ribbon, select the appropriate element.



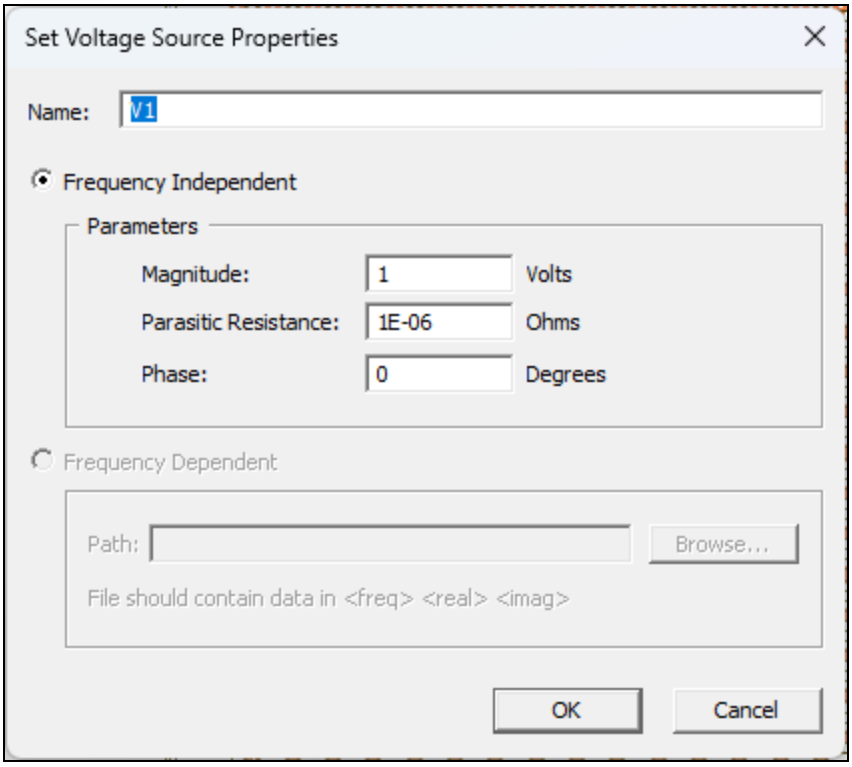
> **Note:** DC terminals can also be attached to pin groups or pins. Select a pin group or pin, then click the **Terminal** icon. A DC terminal is generated on each selected item that does not already have an associated terminal of some kind.

Element properties can be edited in the **Properties** window.

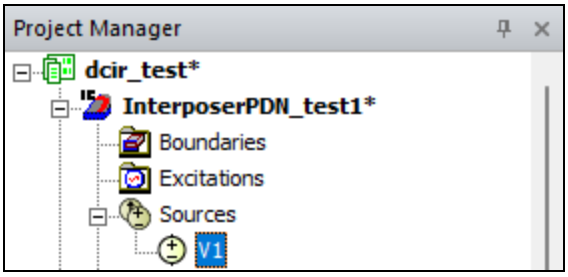


Or right-click the source within the **Project Manager** window and select **Properties** to edit from within the source's respective **Properties** windows.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

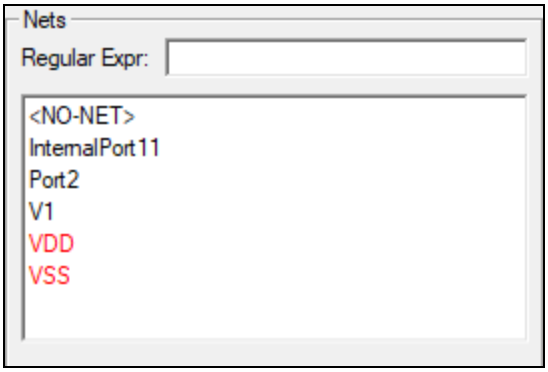


2. Click the location where you want to place the element. The element appears in the **Project Manager** window (i.e., under **DC Terminals** or **Sources**).

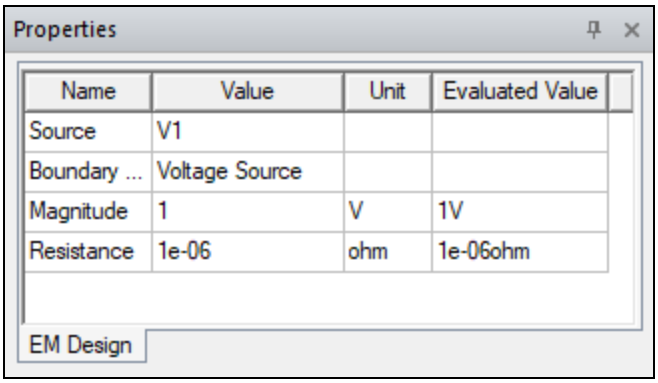


The element also appears in the **Nets** window (e.g., **V1**).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

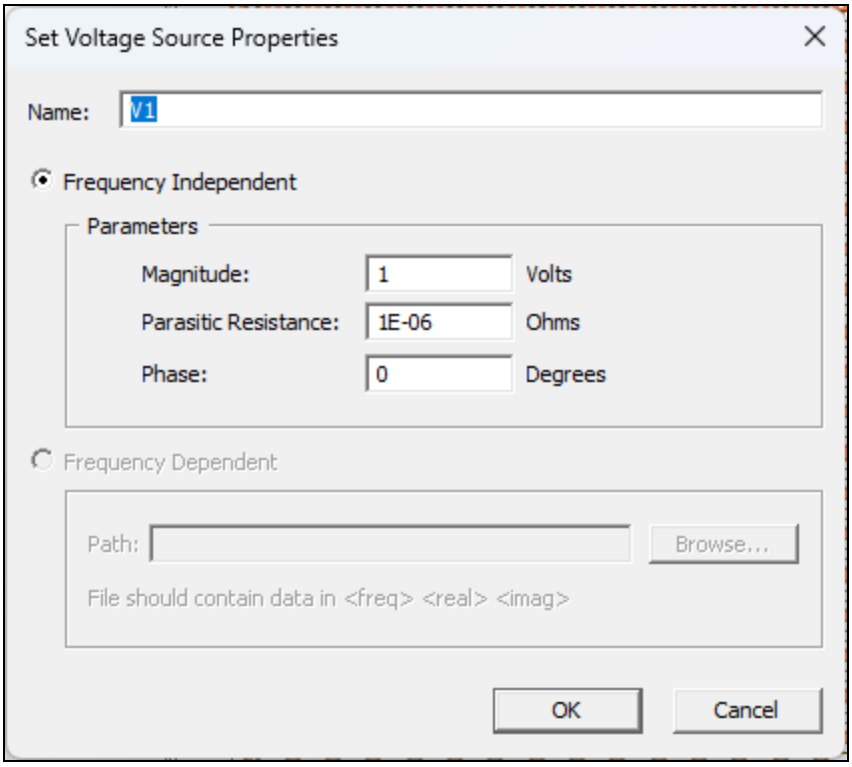


Element properties can be edited in the **Properties** window.



Or right-click the source within the **Project Manager** window and select **Properties** to edit from within the element's respective **Properties** windows.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating Sources on Components

From the **Layout** tab, the following commands are available for managing sources, probes, and terminals. These options allow you to create or remove elements directly on components, ensuring accurate setup for DC IR analysis and hierarchical simulations.

- **Create Current Sources On Component** – Generate current sources attached to the selected component. Refer to DC IR and Icepak Simulation on Hierarchical Designs.
- **Remove Current Sources From Component** – Delete existing current sources from the selected component.
- **Create Voltage Sources On Component** – Generate voltage sources attached to the selected component.
- **Remove Voltage Sources From Component** – Delete existing voltage sources from the selected component.
- **Create Voltage Probes On Component** – Add voltage probes to the selected component for monitoring potential.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Remove Voltage Probes From Component** – Delete voltage probes previously added to the selected component.
- **Create DC Terminals On Component** – Generate DC terminals on pins or pin groups of the selected component.
- **Remove DC Terminals From Component** – Delete DC terminals from the selected component.
- **Create Multiphase VRM** – Refer to Viewing and Creating VRMs.

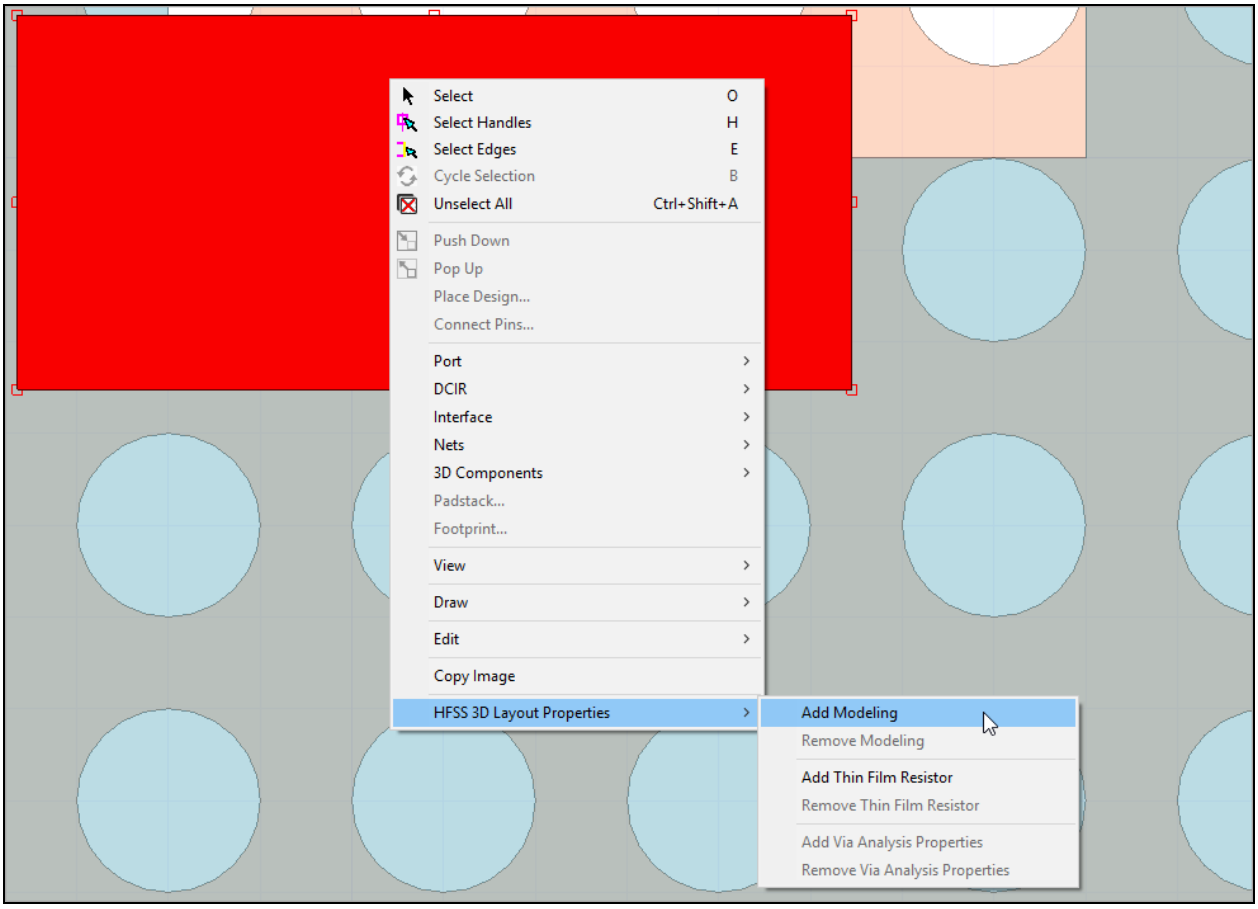
Defining Equipotential Regions (General Mode Only)

The DC IR solver uses equipotential regions to force equal voltage across a specified region. Equipotential regions may be defined as rectangular areas on surface signal layers (i.e., top or bottom). Pads of single-layer vias on surface layers may be used to define an equipotential region.

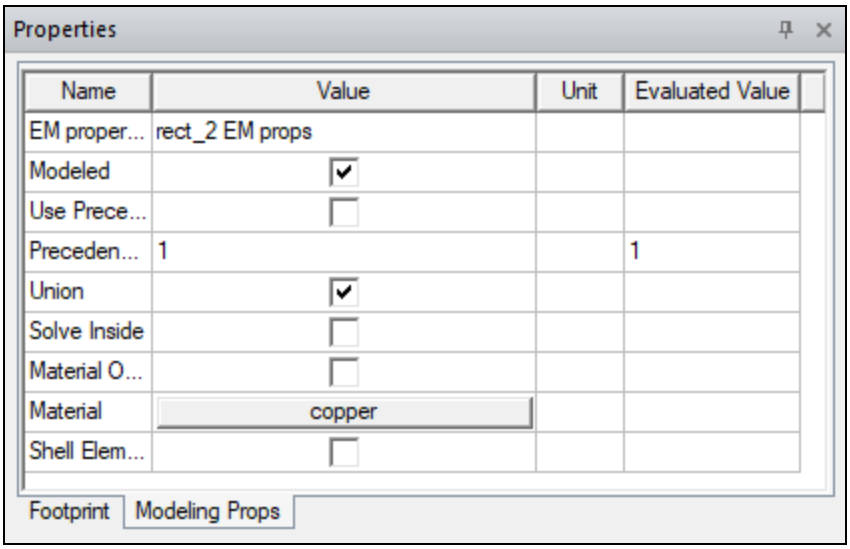
Follow these steps to define equipotential regions.

1. From the **Layers** window, click a surface layer.
2. Create a rectangle or select a single-layer via.
3. With the rectangle or via selected, right-click and select **HFSS 3D Layout Properties** > **Add Modeling**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. From the **Properties** window, click the **Modeling Properties** tab.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

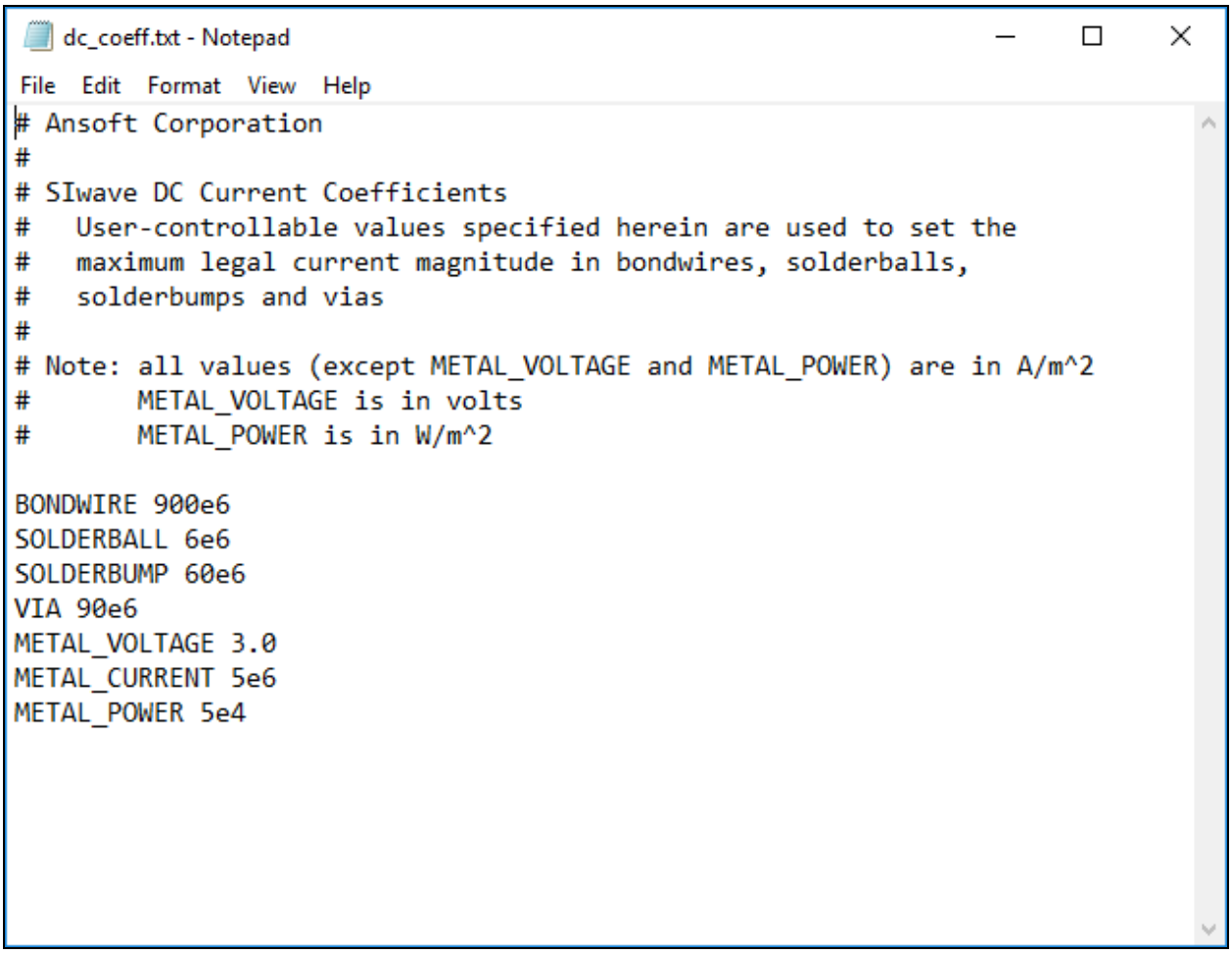
5. From the **SIwave** area, check the **DCIR Equipotential Region** box to define the region as eqipotential.

Editing the DC Coefficient Text File

The DC Coefficient text file (**dc_coeff.txt**) is automatically installed in the <*install_root*>\ANSYS Inc\v261\AnsysEM" directory.

In this file, set the threshold values for bondwires, solderballs, solderbumps, vias, metal voltage, metal current, and metal power.

All values must be positive.



Viewing and Creating VRMs (General Mode Only)

Voltage Regulator Modules (VRMs) supply near-constant DC voltage to subsystems on a board, such as the CPU or memory. Some VRMs make use of remote sense lines to monitor the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

voltage at their load. This allows them to use feedback to maintain a more stable voltage at the point in the power delivery network where current is being drawn, even if that current varies widely depending on the system's activity level. Some VRMs use multiple phases, and have more than one chip (power module) supplying current to the load. This has advantages in terms of component prices, heat management, and failure tolerance. VRMs with multiple phases try to balance the currents across phases and try to keep the current in each phase nearly the same.

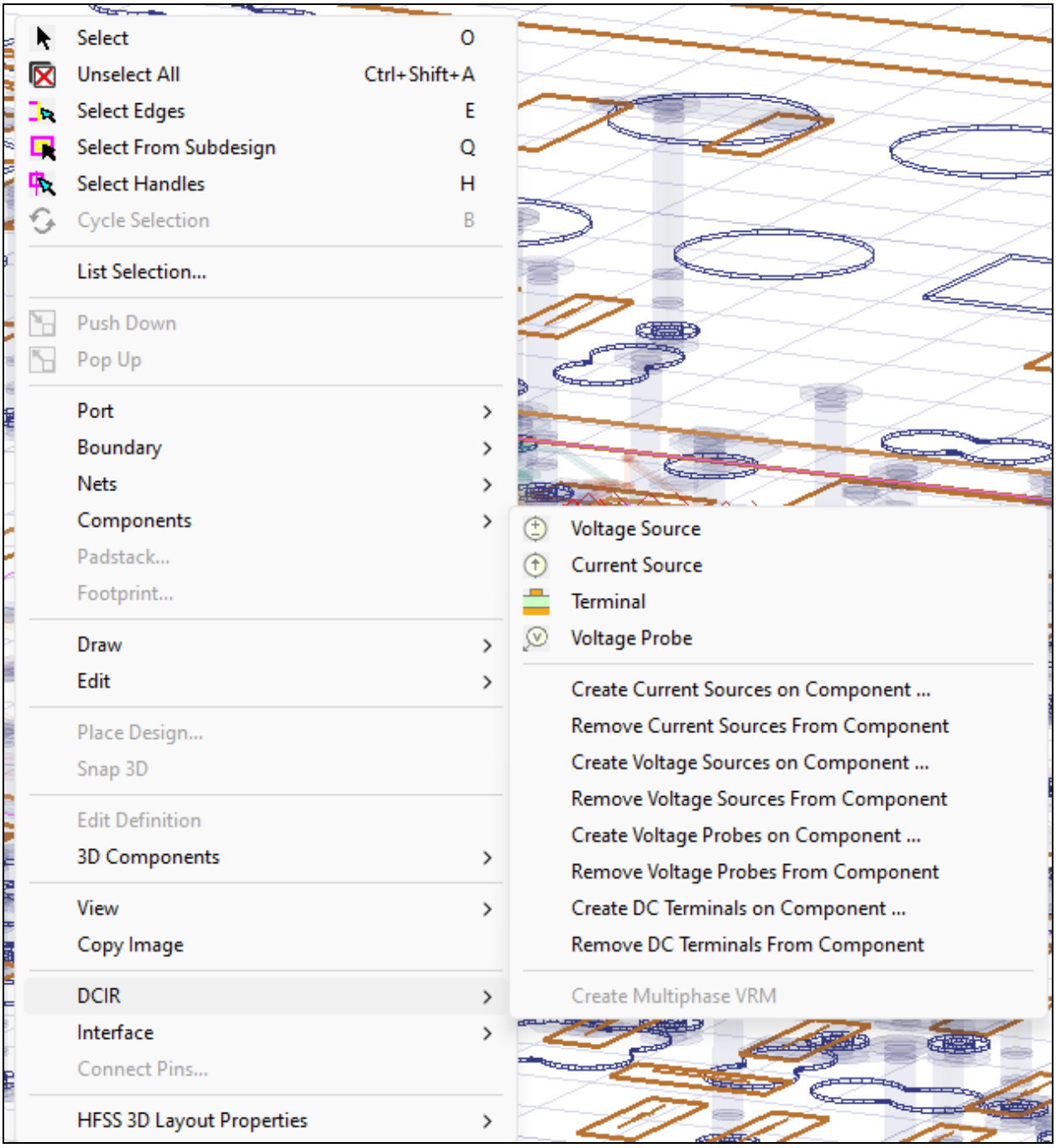
Some VRMs also have a separate chip (control module) that monitors the load voltage and the phase currents and sends out control signals to adjust the output voltage and balance the phase currents. In some designs this control capability is built into the power modules themselves.

You can define multiphase VRMs and run the DC IR simulation. HFSS 3D Layout can compare current and power output at each VRM phase to ensure that values are in balance. These values can also be compared against the simulation results generated by modeling the VRMs using voltage sources.

To view or create Voltage Regulator Modules:

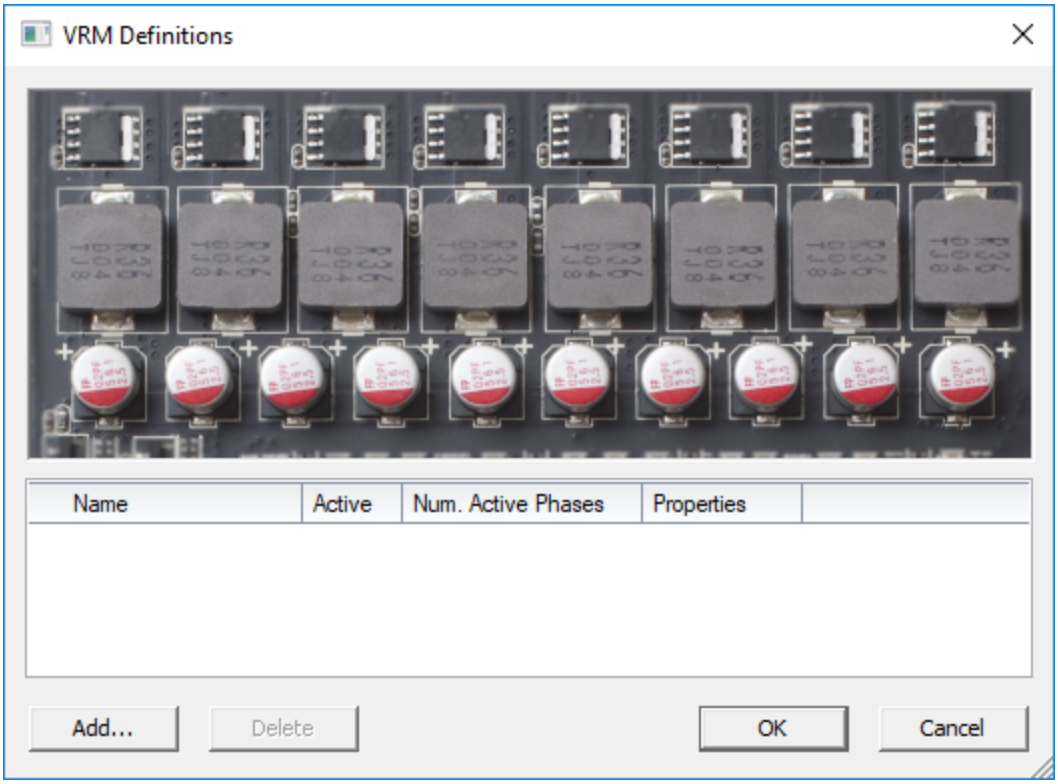
1. Click **Layout**.
2. From the right side of the ribbon, click **VRMs**. Alternatively, right-click in the design area and select **DCIR** > **Create Multiphase VRM**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **VRM Definitions** window opens.

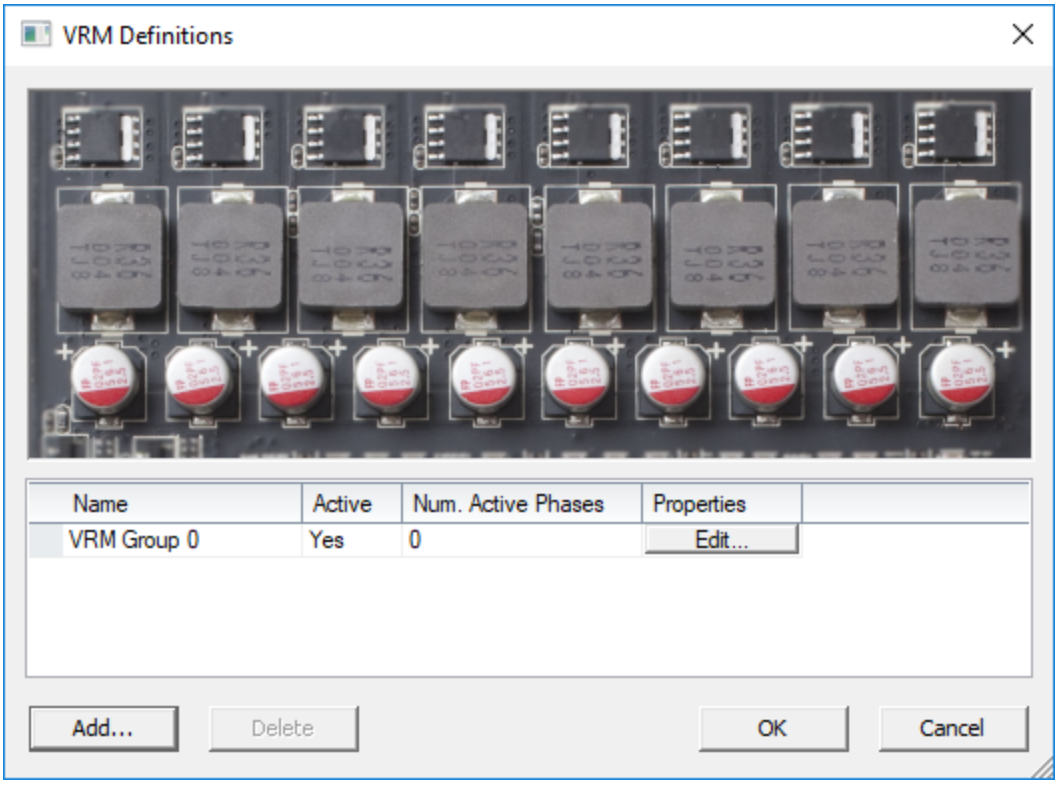
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



VRMs currently in the design are listed.

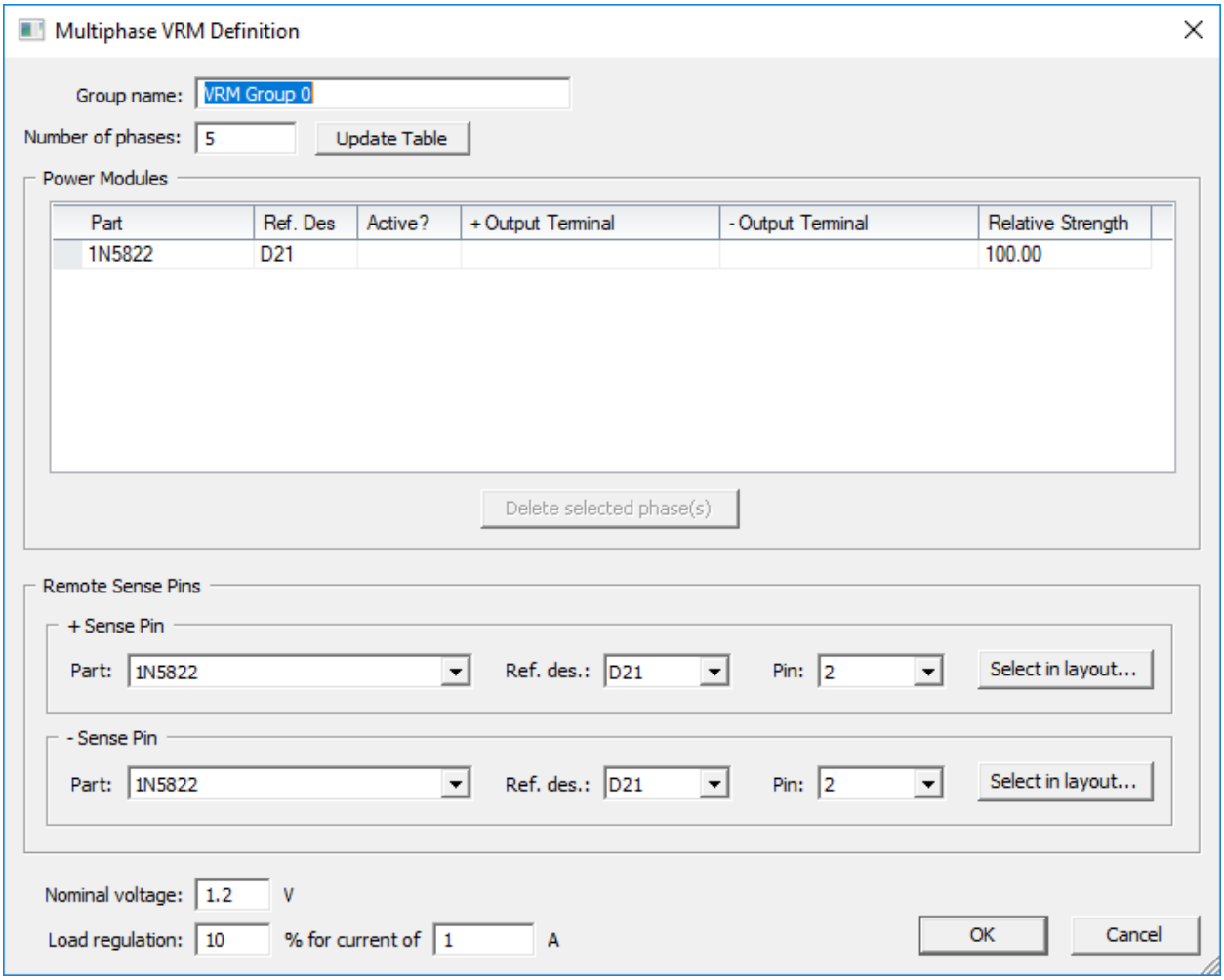
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To add a VRM, click **Add**. A new line appears 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.

4. To define the new VRM, click **Edit** to open the **Multiphase VRM Definition** window.



5. From the **Group name** field, enter a name.

   This group is a collection of ICs that constitutes a multiphase voltage regulator. Typically, it consists of several power module ICs (one per phase) that source the output currents to the load. In some designs there may be a controller IC that monitors the load voltage and phase currents and sends out control signals to the power modules.

6. Specify the **Number of phases**. Phases appear as rows in the Power Modules list.
7. For each phase, click the cells to define the **Part**, **Reference Designator**, **Active/Inactive** status, pin numbers for the **Positive (+) Output Terminal** and **Negative (-) Output Terminal**, and a **Relative Strength**.

   For **Relative Strength**, all power modules are nominally expected to supply the same output current, but in reality has small differences. Manufacturers often specify a current sharing accuracy (example: +/- 10%). Different relative strengths for each phase allow you

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to model such differences. These are expressed as a percentage, with a default value of 100% (exactly matched strength). You can raise or lower these percentages as needed, even dropping them to zero to model a failure of one power module. A strength of 100% is normal strength; a phase with 90% strength supplies only 0.9 times the normal current. The voltage actually produced by the VRM varies on the nominal value depending on how much load current is being drawn and how many phases are used. The variation can be estimated on the load regulation data entered:

Change in V_{out} = (load regulation fraction) x (nominal voltage) x (actual output current) / ((current in load regulation spec) x (number of phases))

8. From the **Remote Sense Pins** area, use the drop-down menus to select the positive and negative remote sense pins. Alternatively, click **Select in layout** and click the pin in the Design Area.

   Each VRM group has remote sense lines to monitor the load voltage. There is both a positive and negative sense line and they are connected to pins of a component. You can specify the part number and reference designator for the control module IC (if a separate one is used; otherwise identify one of the power modules) and the pins that are used for remote voltage sensing.

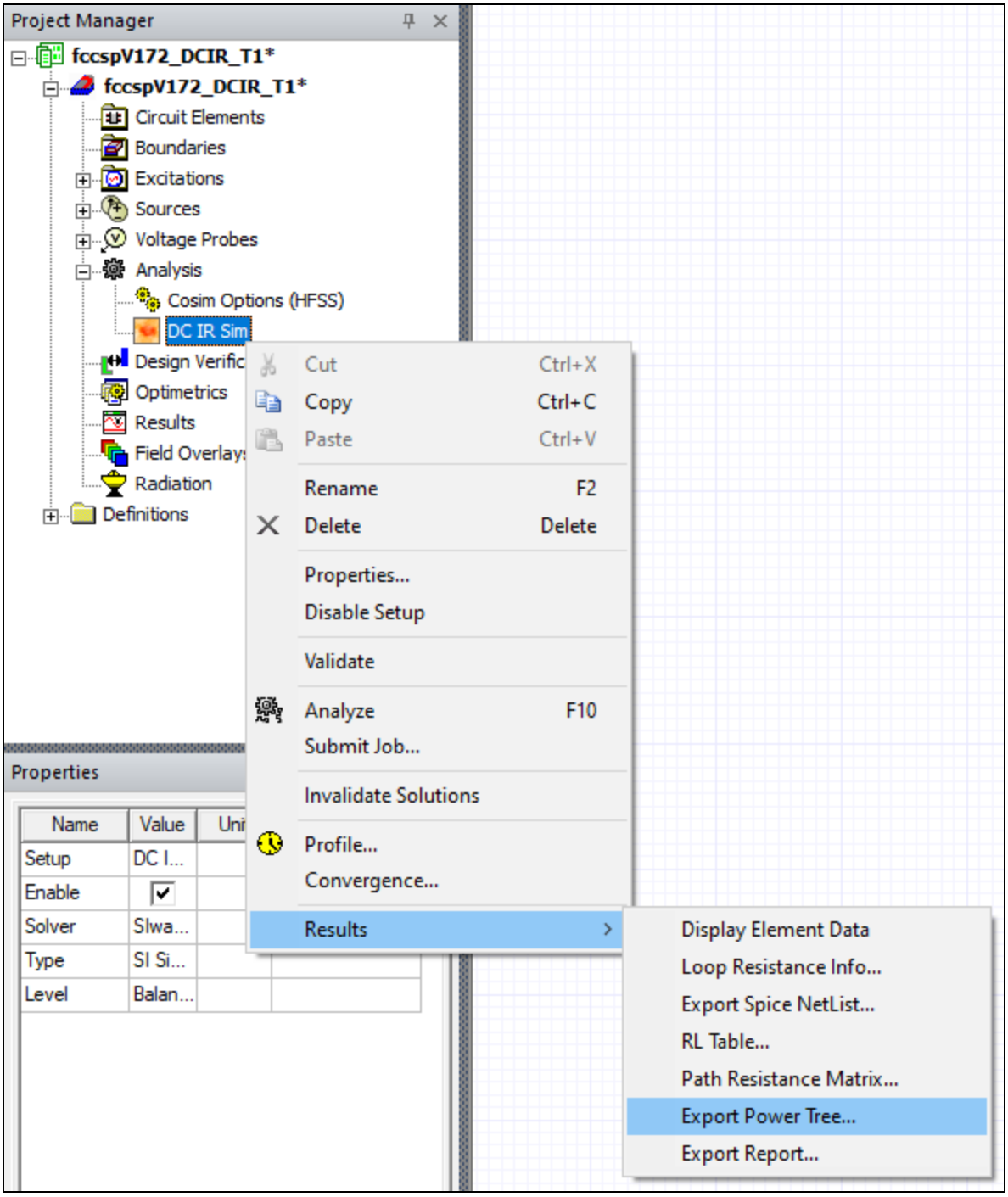
9. If selected, change the values for **Nominal voltage** and **Load regulation**. **Nominal Voltage** is what the output voltage should be with zero load current. It is the selected voltage difference between the positive and negative sense pins; the VRMs adjust their output voltage to meet this value within tolerance.
10. **Load regulation** is how much the output voltage of a single power module varies from its nominal value for a given load current. The voltage variation is entered as a percentage of the output voltage.
11. Click**OK** to save the settings and exit the window.

Viewing and Exporting DC IR Solution Results

Follow these steps to view DC IR results.

1. From the **Project Manager** window, expand **Project Tree** > **[*Active Design Folder*]** > **Analysis**. Then right-click the DC IR simulation (e.g., **DC IR Sim**) and select **Results** > **[*Appropriate Result Type*]** (e.g., **Export Power Tree**).

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 result type you open a settings menu and customize the final output. Options include the following:

- Display Element Data
- Loop Resistance Info

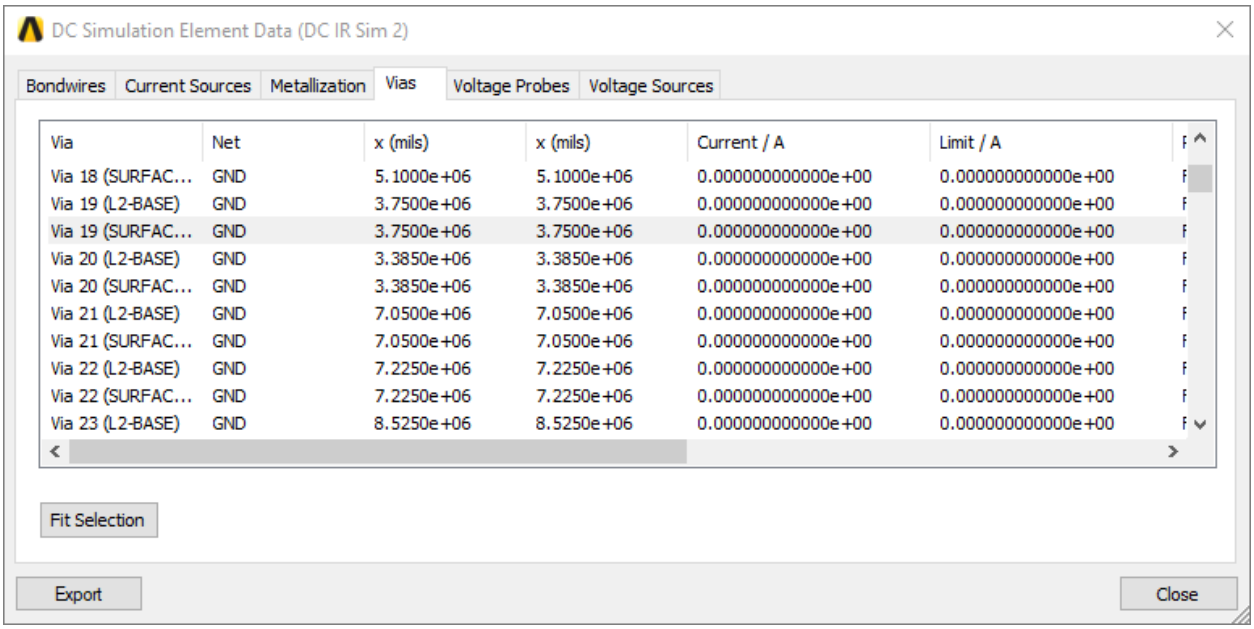
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Export Spice NetList
- RL Table
- Path Resistance Matrix
- Export Power Tree (**General Mode Only**)

Display Element Data

Follow these steps to view and export DC IR data.

1. From the **Project Manager** window, expand **Project Tree** > **[*Active Design Folder*]** > **Analysis**. Then right-click the DC IR simulation (e.g., **DC IR Sim**) and select **Results** > **Display Element Data** to open the **DC Simulation Element Data** window.



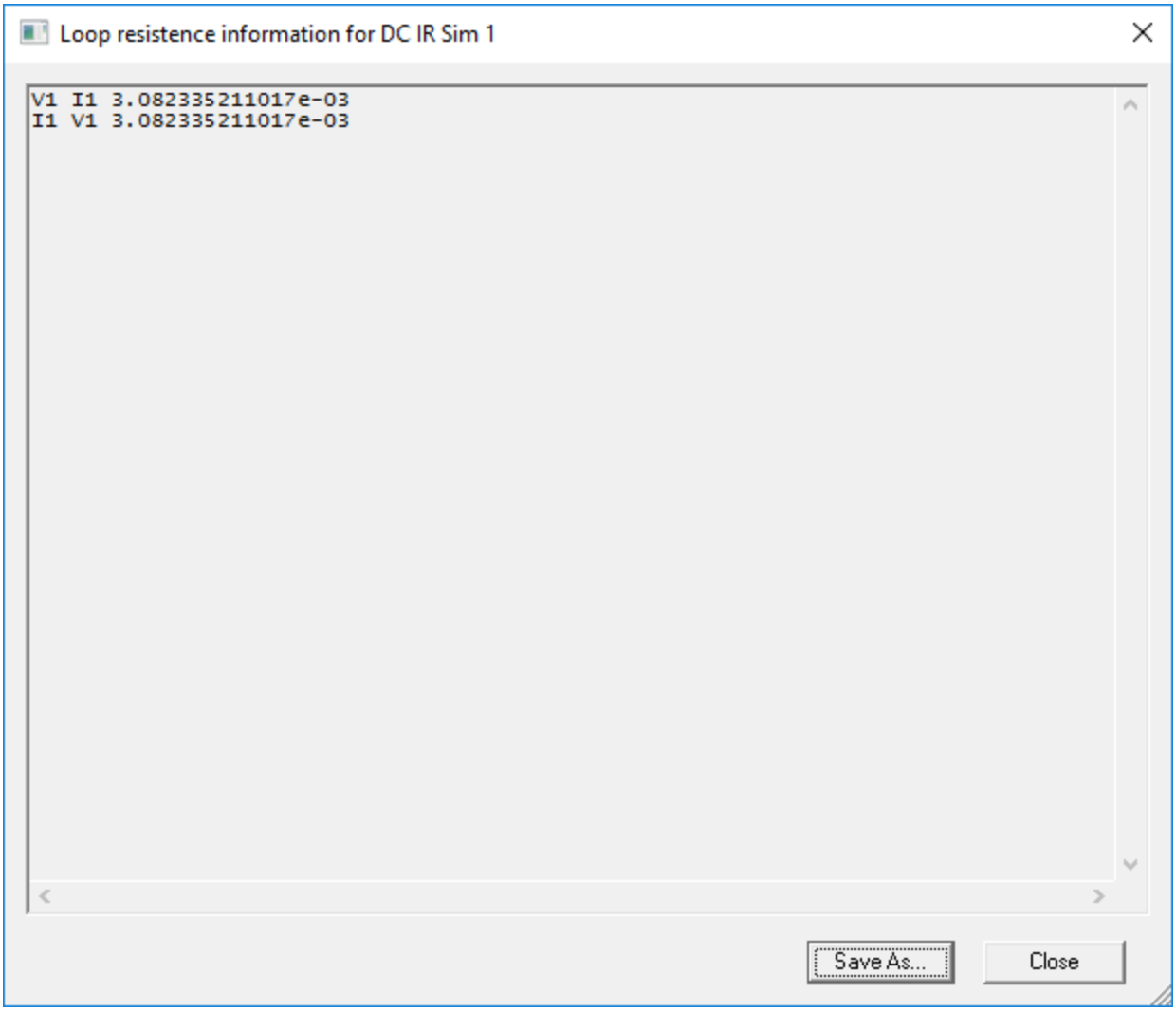
2. Select any element to highlight the element in the design area and populate the Properties window with relevant information. **Ctrl**+**click** to select more than one element at a time or **Ctrl**+**A** to select all.
3. Click **Fit Selection** to focus and zoom in on a selected element.
4. Use the tabs to move between elements, and click column headers to sort the data. The **Pass/Fail** column shows **Pass** if the total current is less than or equal to the current limit flowing through the element.
5. Click **Export** to save the data as a DC Element Data File (*.dc).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Loop Resistance Info

The loop resistance is computed per pair of sources where the first source is the location of the measurement, and the second is the return path of the current loop. Note the source used as return path is shorted and the internal resistance is ignored. Follow these steps to view loop resistance information in Ansys SIwave™ printed circuit board (PCB) and package electromagnetics simulation software.

1. From the **Project Manager** window, expand **Project Tree** > **[*Active Design Folder*]** > **Analysis**. Then right-click the DC IR simulation (e.g., **DC IR Sim**) and select **Results** > **Loop Resistance Info** to open the **Loop resistance information** window.



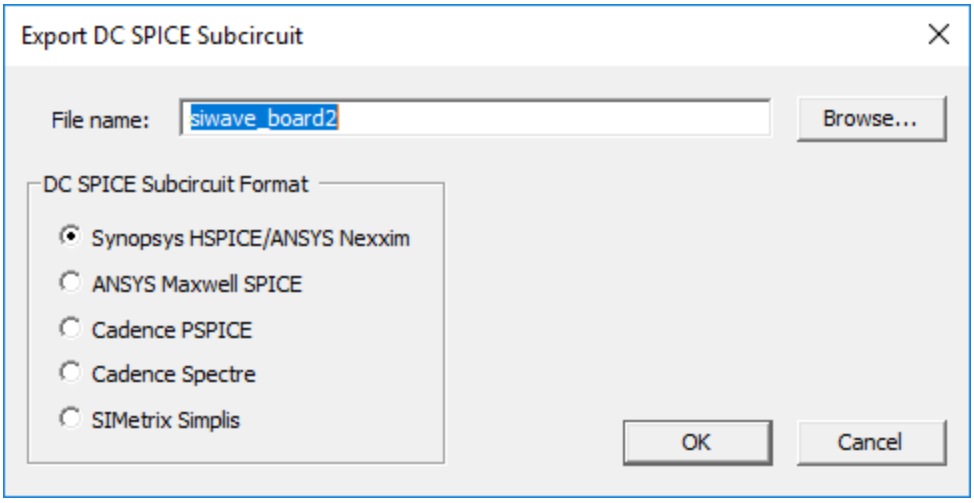
2. Click **Save As** to save the Loop Resistance information in plain text format.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export DC SPICE Subcircuit

Exporting a subcircuit allows you to perform an analysis of the structure in a circuit simulator, together with detailed nonlinear models of the drivers and receivers. Also simulate systems comprising PCBs and packages by connecting their subcircuits together in SPICE or Spectre. Follow these steps to export a SPICE Subcircuit.

1. From the **Project Manager** window, expand **Project Tree** > **[*Active Design Folder*]** > **Analysis**. Then right-click the DC IR simulation (e.g., **DC IR Sim**) and select **Results** > **Export Spice NetList** to open the **Export DC SPICE Subcircuit** window.



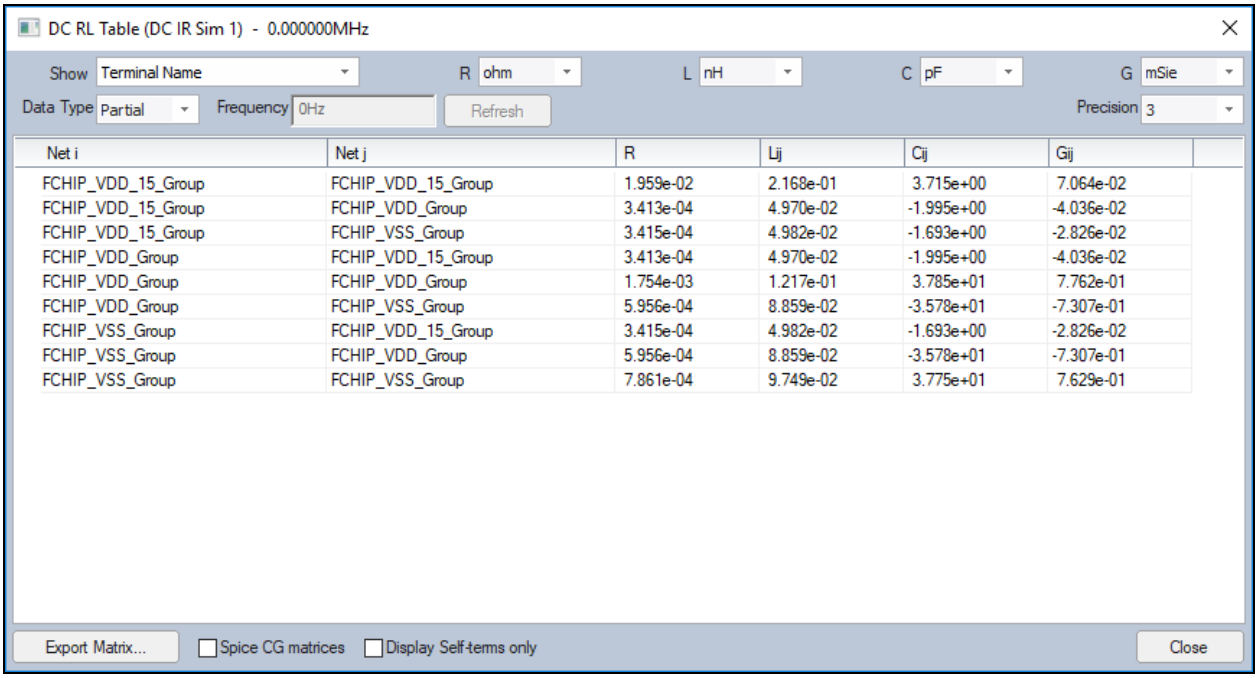
2. **Browse** to the selected directory to save your file.
3. Select the appropriate SPICE format.
4. Click **OK**.

RL Table

The RL Table displays RLCG values for terminals and nets. Follow these steps to view the RL Table.

1. From the **Project Manager** window, expand **Project Tree** > **[*Active Design Folder*]** > **Analysis**. Then right-click the DC IR simulation (e.g., **DC IR Sim**) and select **Results** > **RL Table** to open the **DC RL Table** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

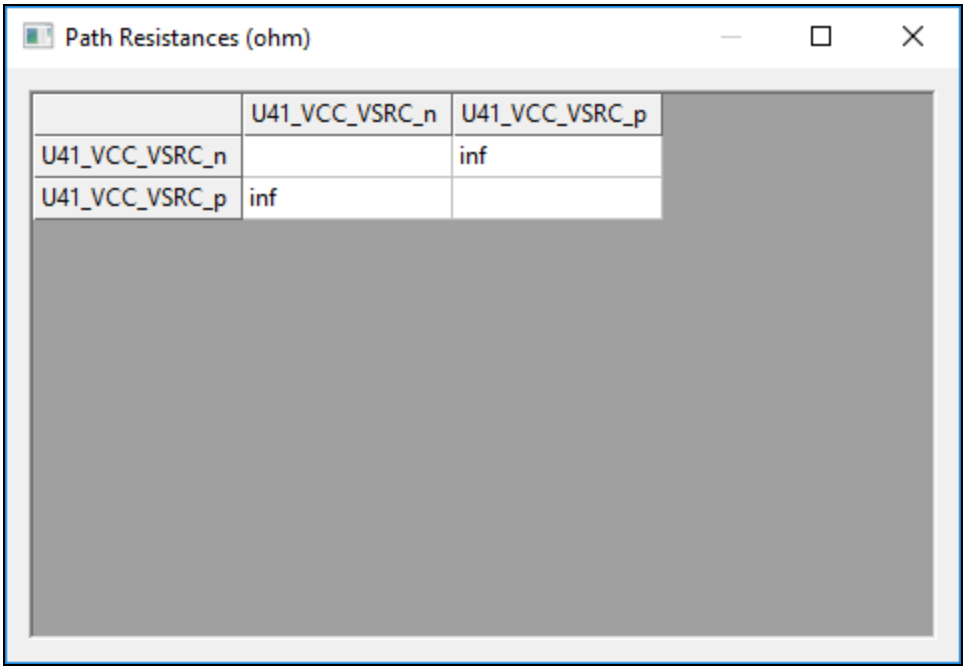


2. Use the drop-down menus to adjust data types, units of measure, and precision for display.
3. Click **Export Matrix** to save the table in either CSV or plain text format.

Path Resistance Matrix

The Path Resistance Matrix shows resistances between pairs of terminals. To view the path resistance matrix, navigate to the **Project Manager** window. Then expand **Project Tree** > **[*Active Design Folder*]** > **Analysis** and right-click the DC IR simulation (e.g., **DC IR Sim**). Select **Results** > **Path Resistance Matrix** to open the **Path Resistances** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

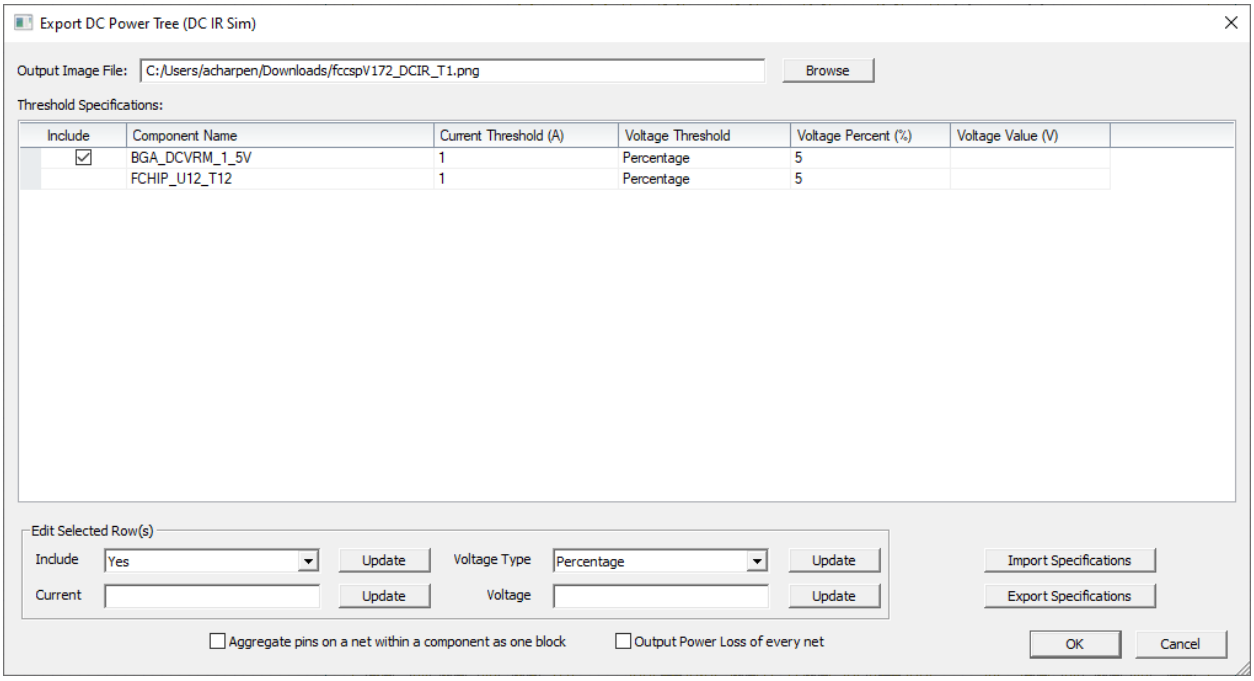


Export DC Power Tree (General Mode Only)

The Power Tree provides a graphical representation of actual voltage, actual current, IR drop, specification, and pass/fail status. Follow these steps to export the Power Tree.

1. From the **Project Manager** window, expand **Project Tree** > **[*Active Design Folder*]** > **Analysis**. Then right-click the DC IR simulation (e.g., **DC IR Sim**) and select **Results** > **Export Power Tree** to open the **Export DC Power Tree** 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. Click **Browse** to select a name and location for your file.
3. Adjust the settings as appropriate.

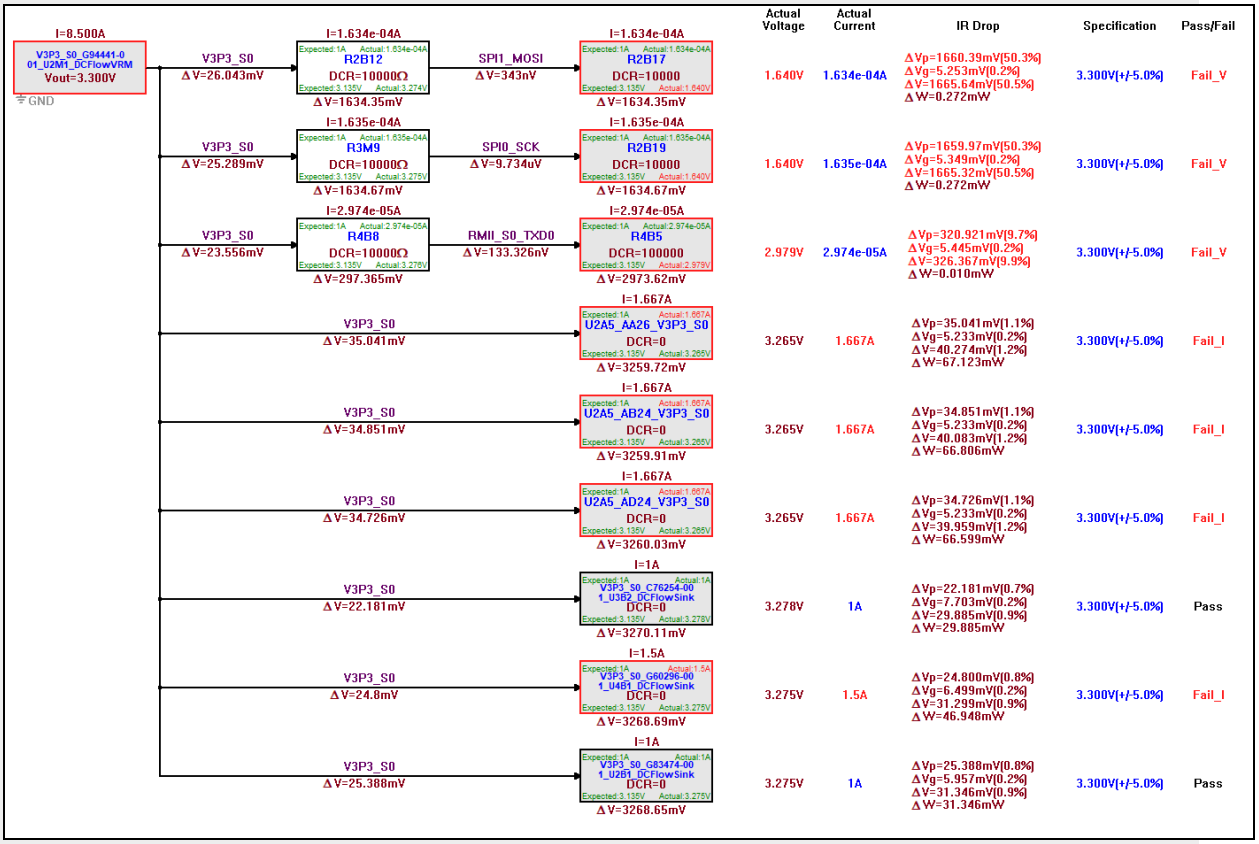
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Beneath the **Edit Selected Row(s)** area, there are two additional options:

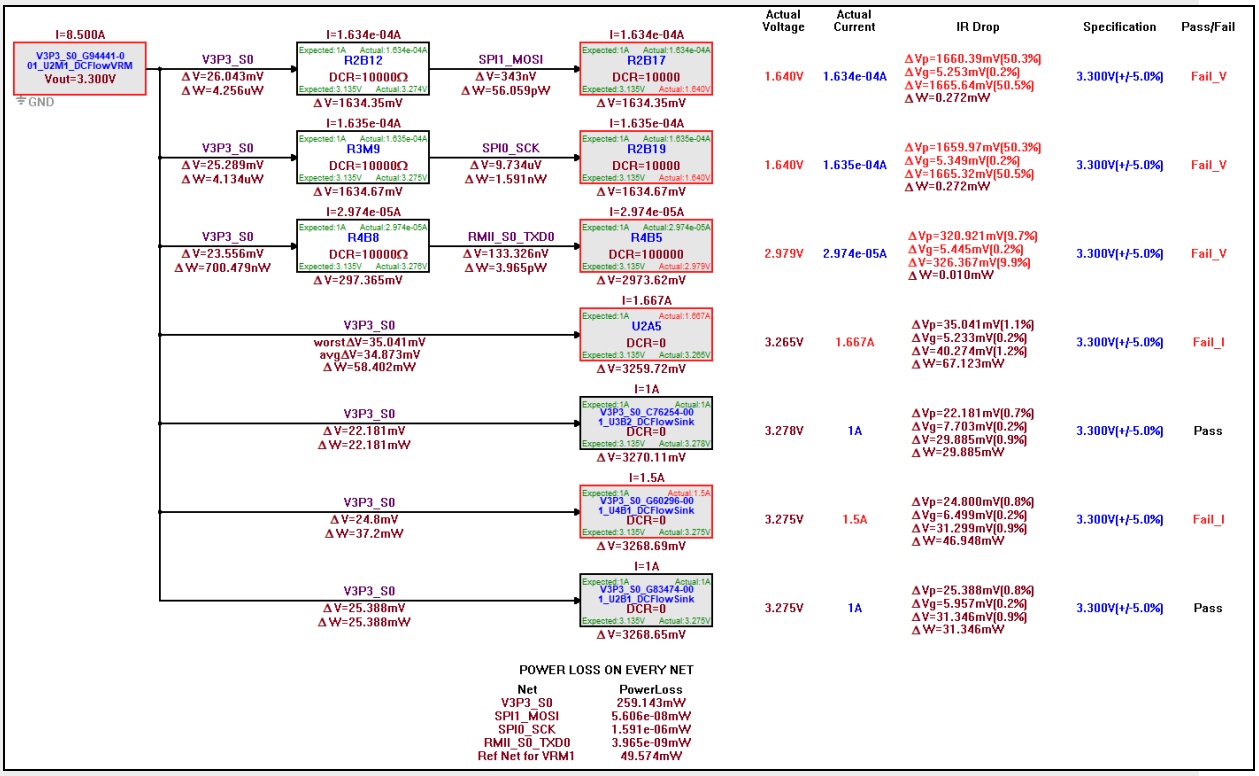
- **Aggregate pins on a net within a component as one block** — check this box if the source/probe is a distributed current, to combine all of the pins from a single component into one block for general analysis. The result will display the combined pins' average voltage loss as well as the most significant voltage loss experienced by any of the pins.
- **Output Power Loss of every net** — check this box to add a list beneath the power tree of every net and its power loss.

Before Checking Either Box



After Checking Both Boxes

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 save the power tree in PNG format and open it in the default image viewer.

Plotting DC IR Field Overlays

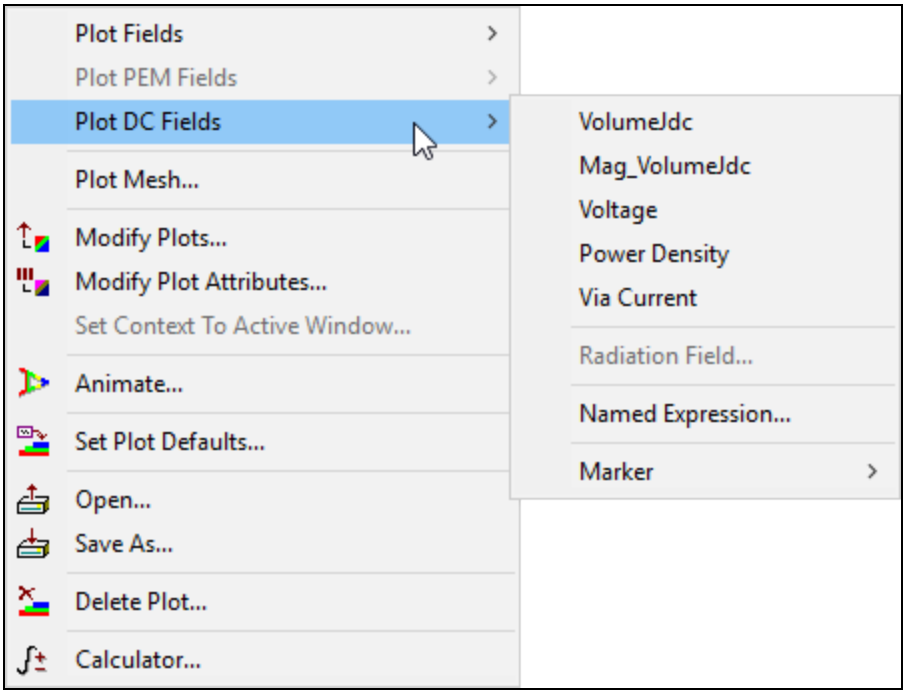
After computing a DC IR Solution, follow these steps to view DC IR field plot overlays.

Note:

All simulation results are available to view and export from General mode and IC mode, except the DC Power Tree. The DC Power Tree can only be accessed and exported from General mode.

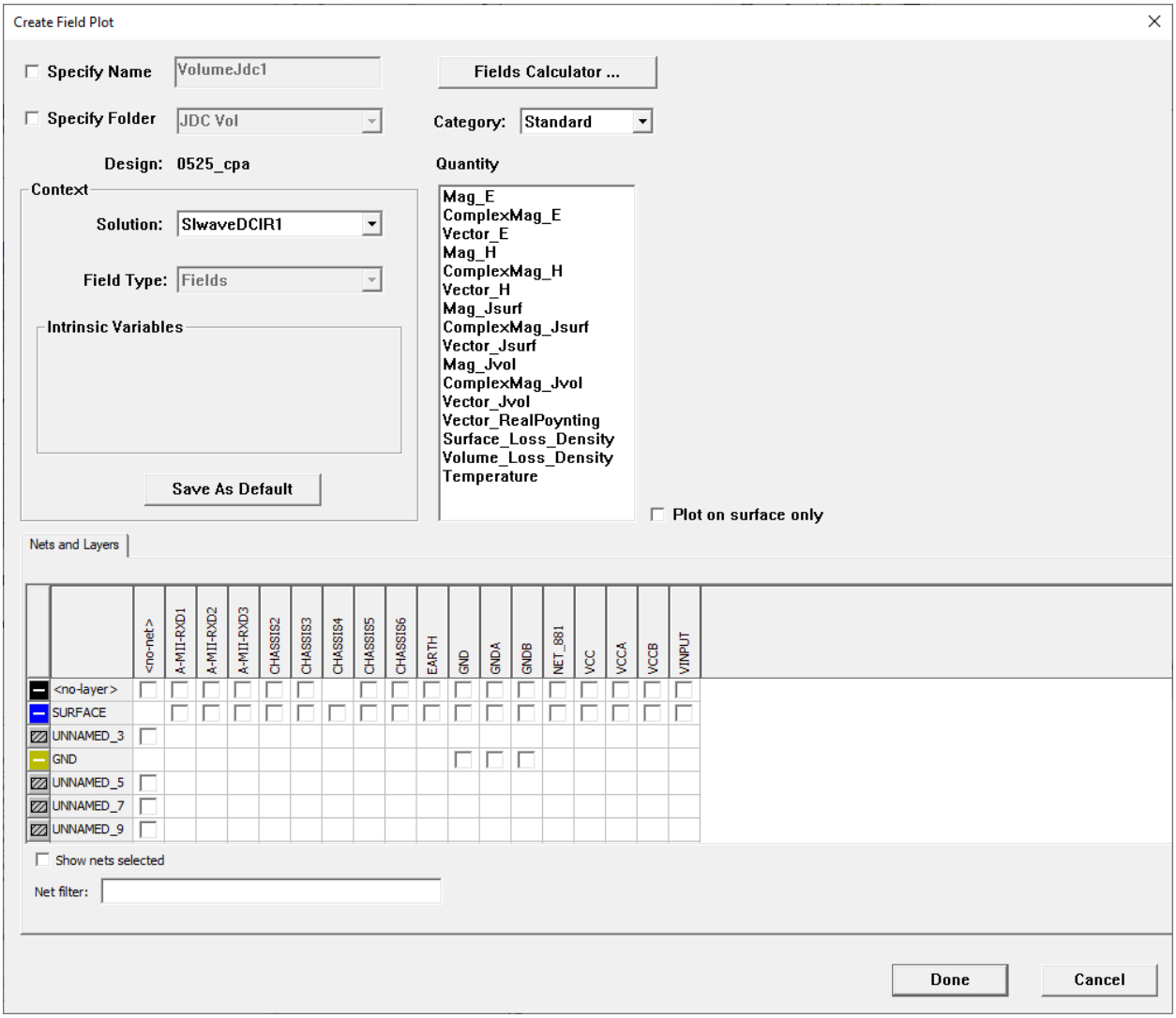
1. From the **Project Manager** window, expand the **Project Tree** and right-click **Field Overlays**. Then select **Plot DC Fields**.

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 overlay type you want to view. Menu options include the following:
   - **VolumeJdc** – overlays the DC current density vector (in Amps/meter^2).
   - **Mag_VolumeJdc** – overlays the Magnitude of VolumeJdc.
   - **Voltage** – overlays voltage.
   - **Power Density** – overlays power density.
   - **Via Current** – overlays current direction on vias.

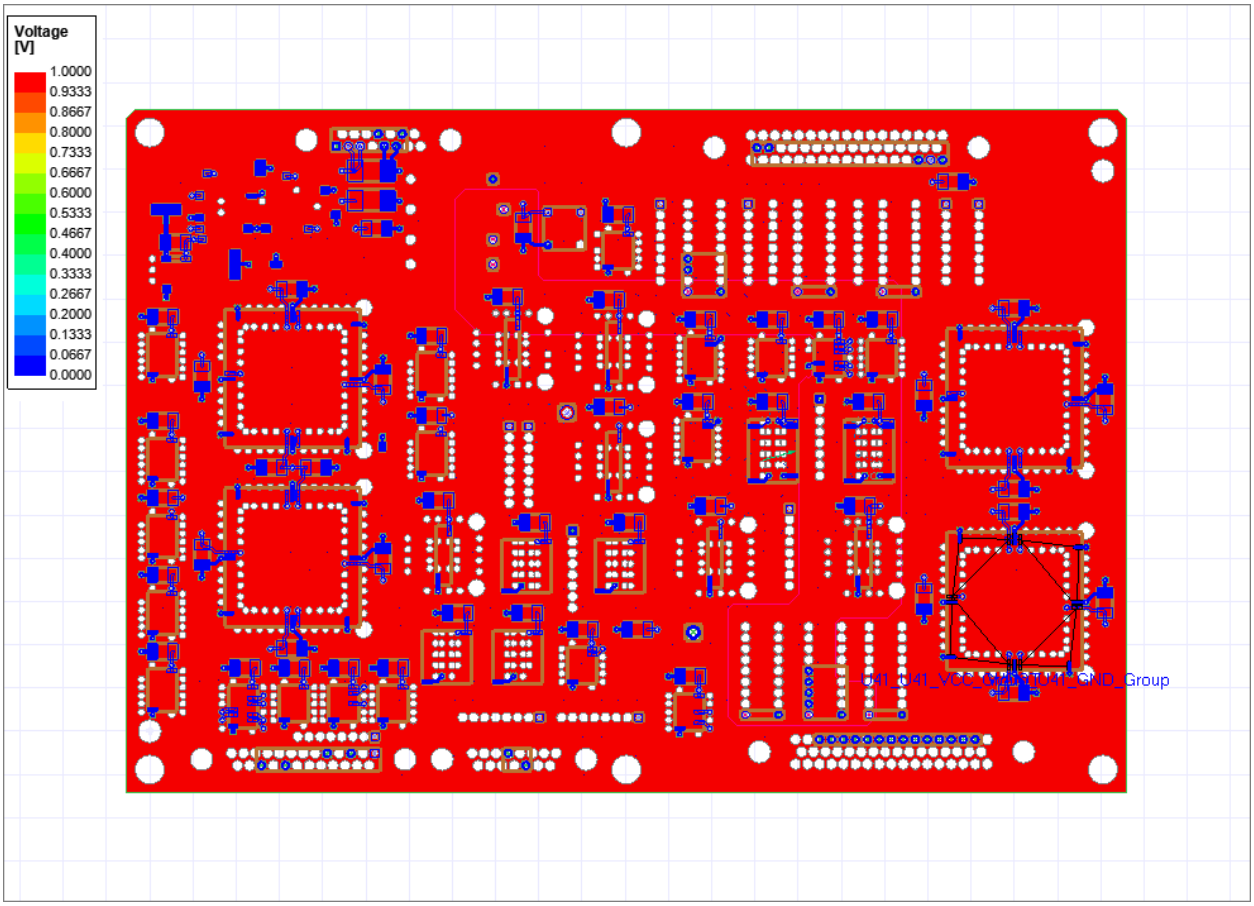
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Specify details or use the Fields Calculator, as appropriate.

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** and the selected field plot overlay appears in the **Layout Editor**.



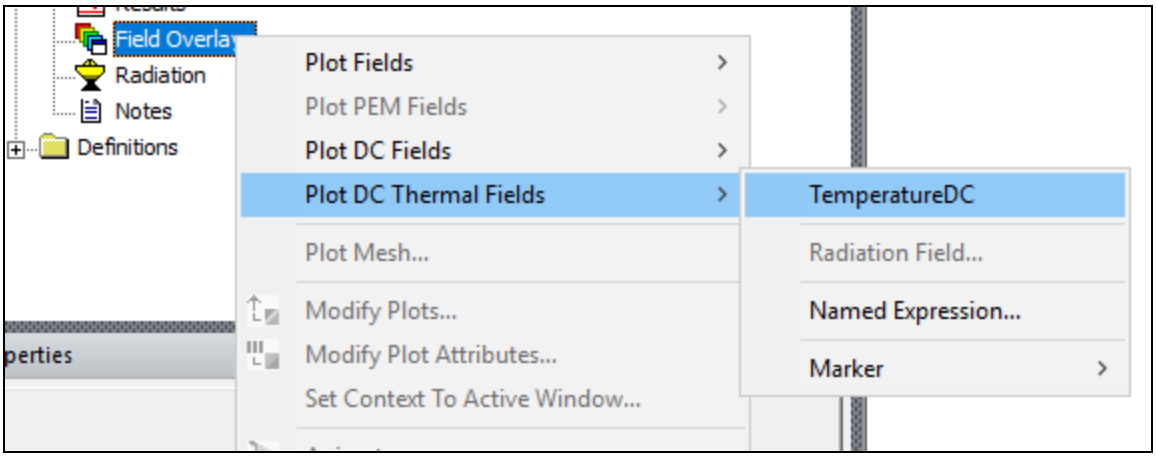
For more general information, see Plotting Field Overlays.

Overlaying DC Temperature Fields (General Mode Only)

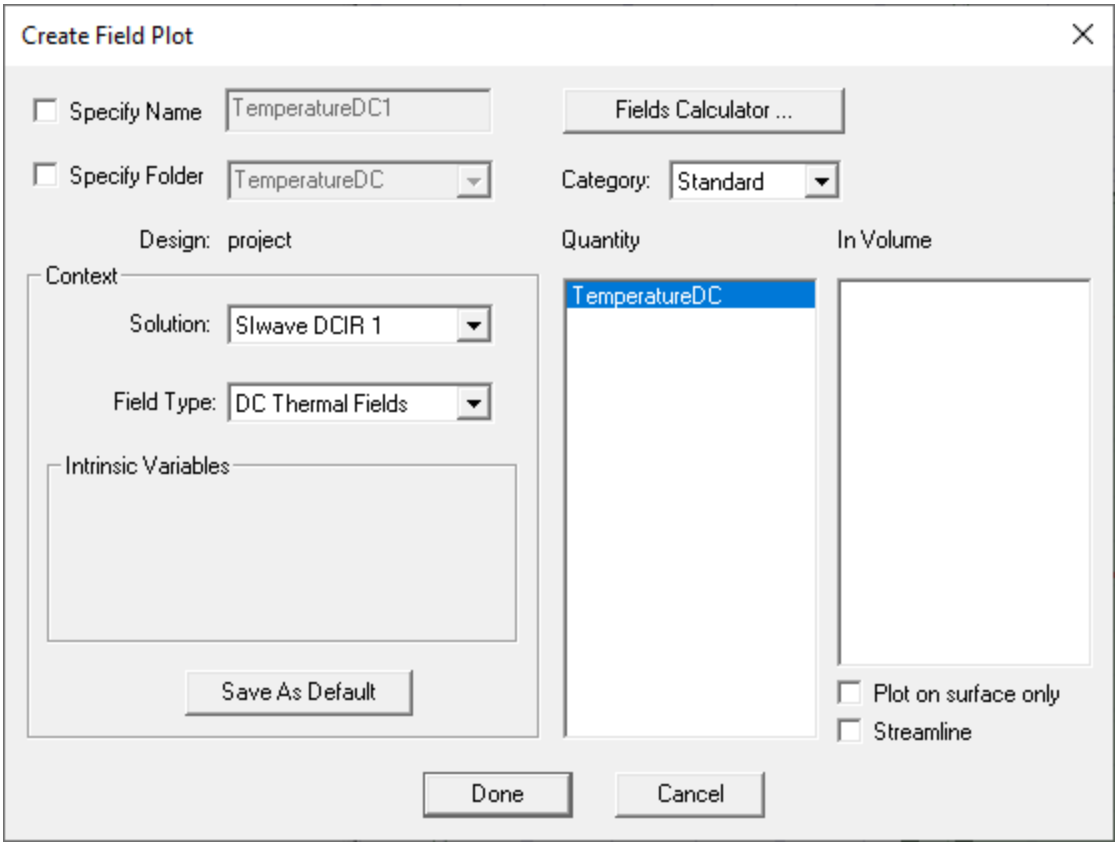
Projects with two-way coupling between Icepak and HFSS 3D Layout can have a field overlay that displays the DC temperatures. The 3D Layout project must have an Icepak DC IR analysis. To create such an overlay:

1. In the Project Manager, open the 3D Layout design and right click **Field Overlays** > **Plot DC Thermal Fields** > **TemperatureDC**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



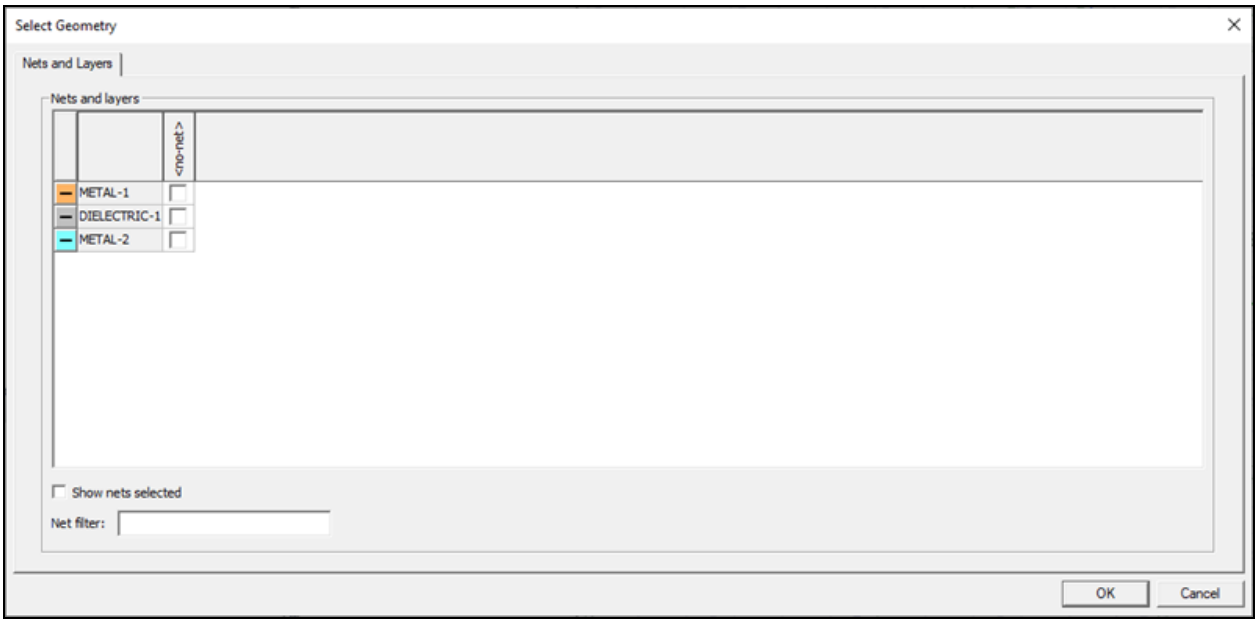
The **Create Field Plot** window opens.



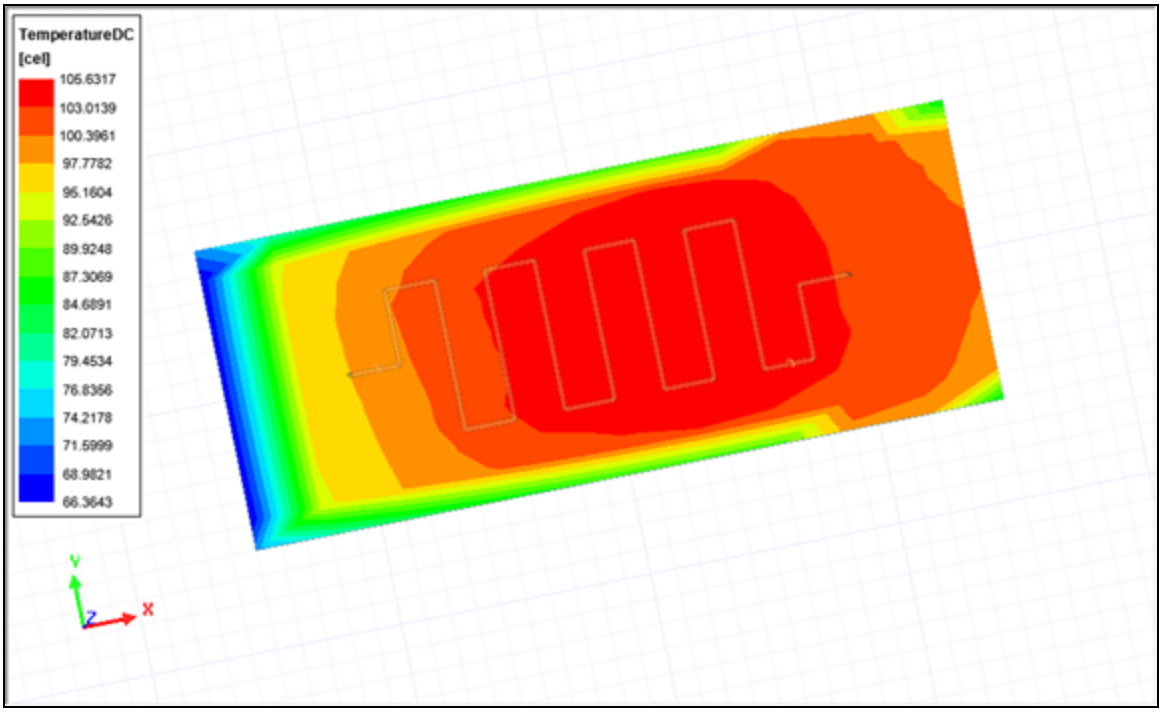
TemperatureDC is automatically selected from the Quantity list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Done**. The **Create Field Plot** window closes, while the **Select Geometry** window opens to its **Nets and Layers** tab.



3. Select the layer whose temperature field should display.
4. Click **OK** and the overlay appears.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

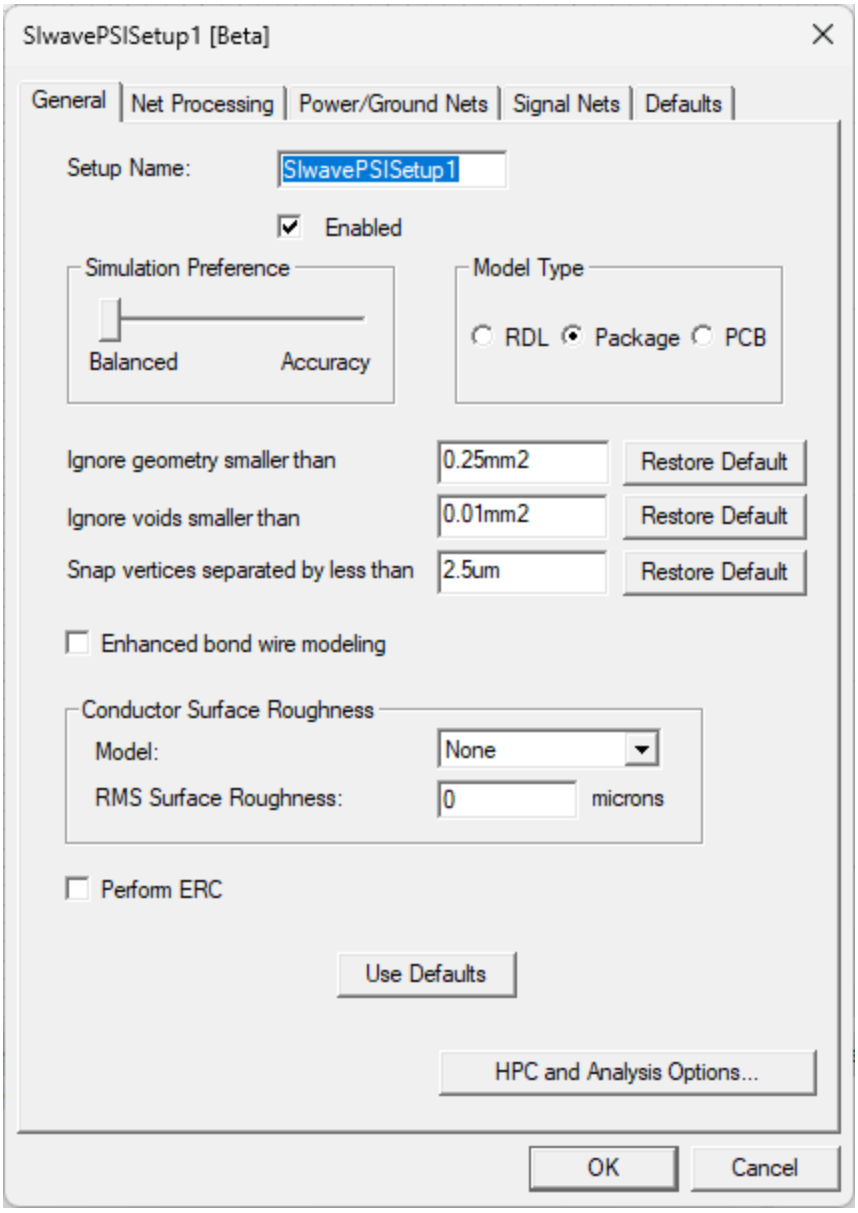
Adding an SIwave-PSI Solution Setup

This section provides step-by-step instructions for configuring an SIwave-PSI solution setup within either the General or IC mode environments.

> **Note:**
>
> The SIwave-PSI Setup is a beta feature in both General and IC modes, although some navigation and customization options are unique to either mode. Refer to Activating the SIwave-PSI Solution Beta Feature for instructions.

1. Open the **SIwavePSISetup1** window. There are several ways to do this, depending on the current environment. For example:
   - From the **Project Manager** window, expand the **Project Tree** > [***Active Design Folder***]. Then right-click **EMDesign1** or **Analysis** and select **Add SIwave Solution Setup** > **Add SIwave-PSI Solution Setup**.
   - From **HFSS 3D Layout**, select **Solution Setup** > select **Add SIwave Solution Setup** > **Add SIwave-PSI Solution Setup**.
   - From the **Simulation** tab, click **SIwave-PSI**.
2. From the **Setup** window, make selections from the **General**, **Net Processing**, **Power/Ground Nets**, **Signal Nets**, and **Defaults** tabs.

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 **General** tab, set the following:
    - **Setup Name** – type a name in the field or accept the default
    - **Enabled** – check the box to activate or deactivate the setup
    - **Simulation Preference** – use the slider to set the simulation to **Balanced** (i.e., use less memory) or **Accuracy** (i.e., use more memory)
    - **General Mode Only:**

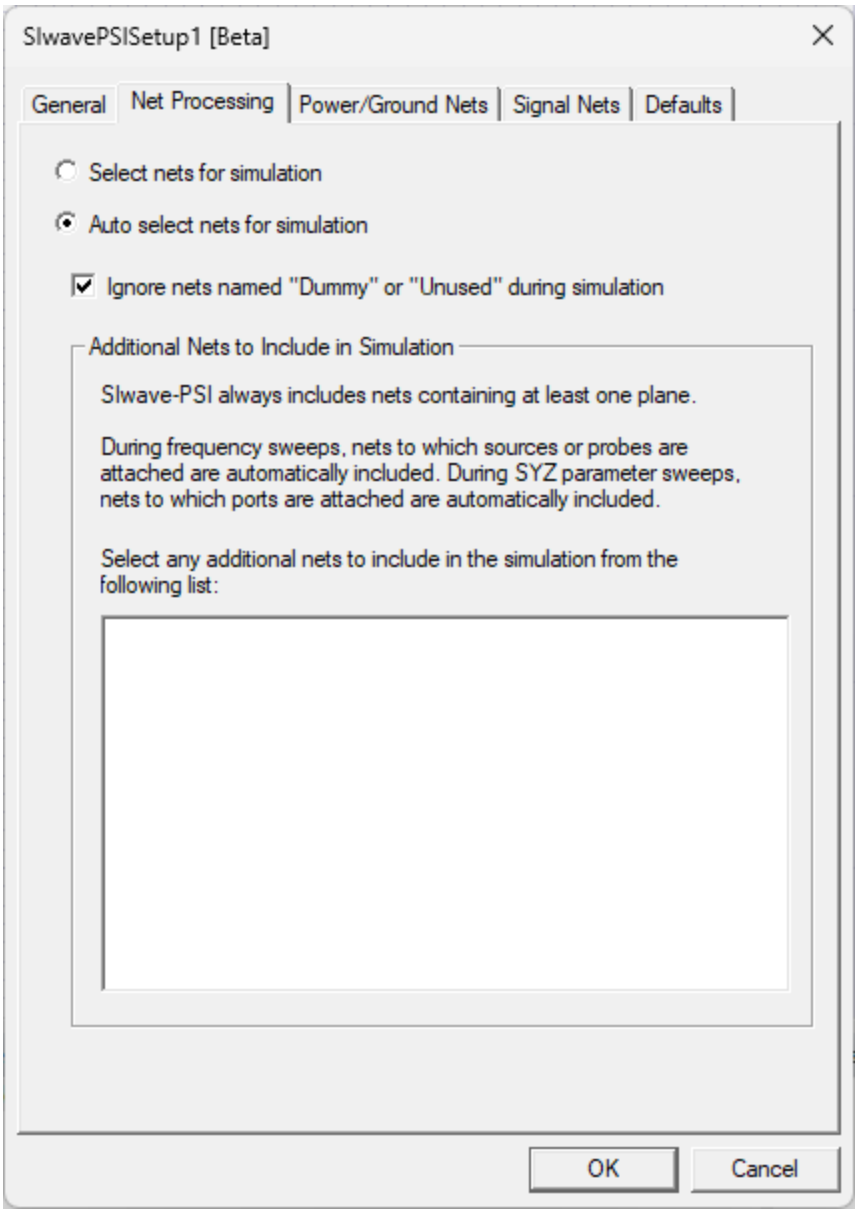
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Model Type** – select **RDL**, **Package**, or **PCB**.
  - **PCB** – for full board layout.
  - **Package** – for physical enclosure and interconnects.
  - **RDL** – for integrated circuit or interposer-level analysis.
- **Ignore geometry smaller than** – enter a value and specify the unit. If applicable, you can **Use Defaults**.
- **Ignore voids smaller than** – enter a value and specify the unit. If applicable, you can **Use Defaults**.
- **Snap vertices separated by less than** – enter a value and specify the unit. If applicable, you can **Use Defaults**.
- **Enhanced bond wire modeling** – check the box to activate. This can slow simulation.

- **Conductor Surface Roughness** area – select a roughness model type from the drop-down menu (i.e., **None**, **Exponential**, or **Hammerstad**) and enter a variable in the **RMS Surface Roughness**field to specify the surface in microns.
- **General Mode Only: Perform ERC** – check the box to determine whether to perform error checking while generating the solver input file.
- **HPC and Analysis Options** – open a window allowing you to specify HPC 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. Click the **Net Processing** tab.



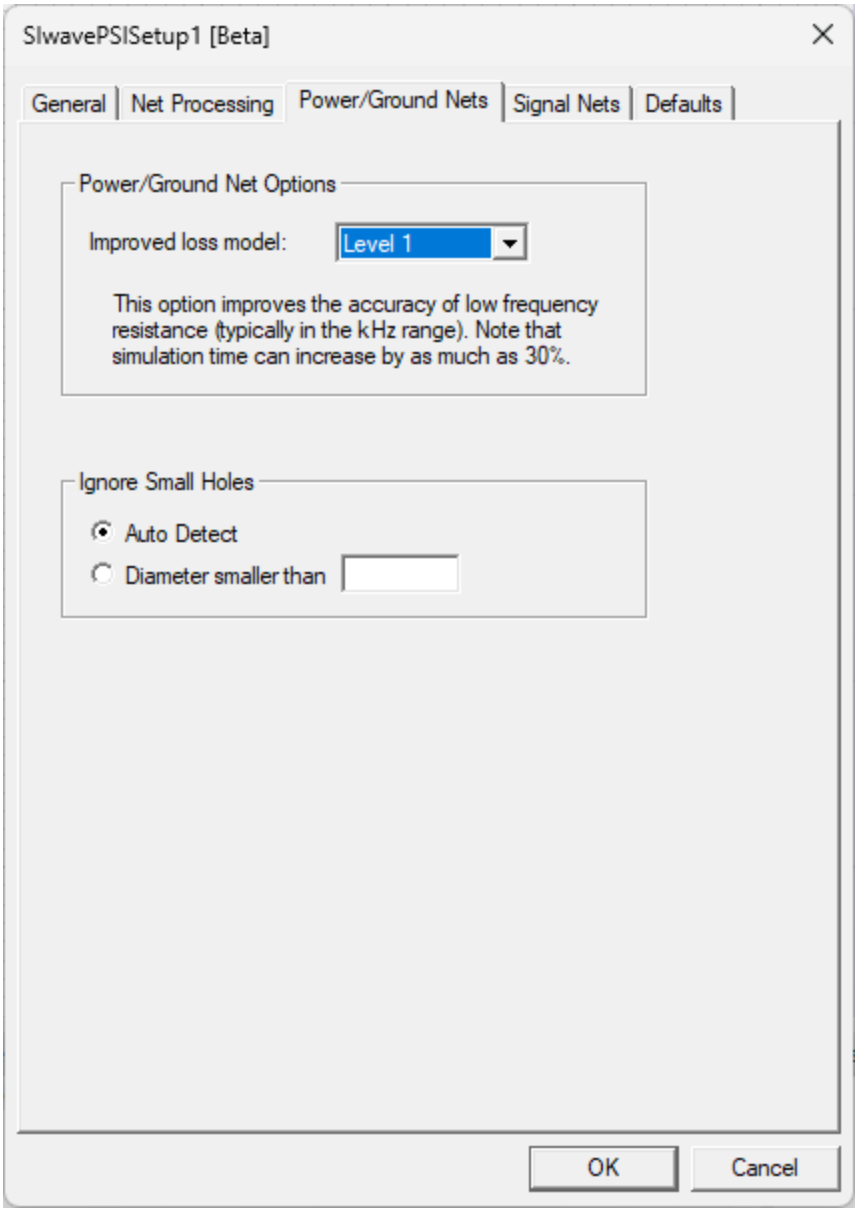
5. From the **Net Processing** tab, set the following:

   - Select **Select nets for simulation** (i.e., allows you to manually select nets to include in the simulation) or **Auto select nets for simulation** (i.e., allows HFSS 3D Layout to select nets to include in the simulation)
   - **Ignore Nets Named "Dummy" or "Unused" during simulation** – check the box to ignore dummy nets

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Additional Nets to Include in Simulation** – when **Auto select nets for simulation** is selected, select additional nets that **must** be included

6. Click the **Power/Ground Nets** tab.



7. From the **Power/Ground Nets** tab, set the following:

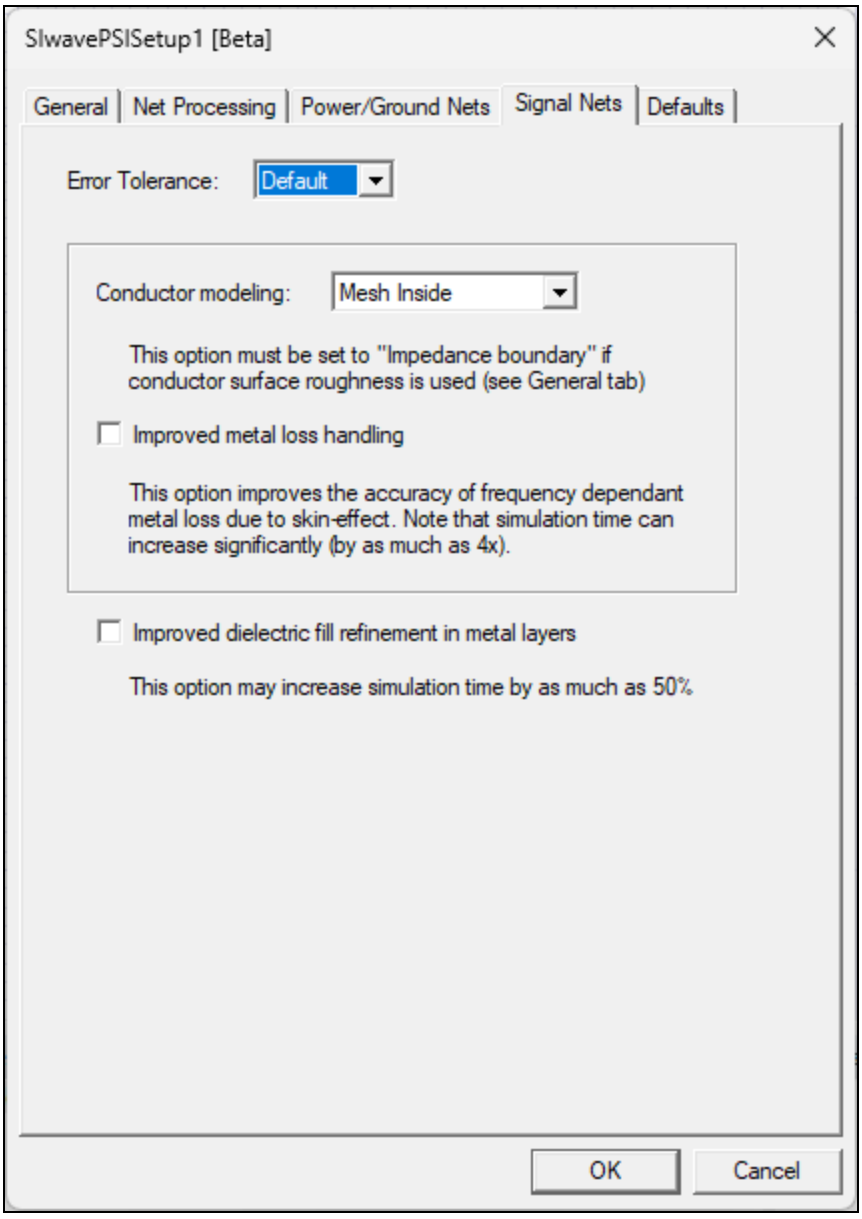
- **Improved loss model** – select an accuracy level from the drop-down menu (i.e., **Level 1**, **Level 2**, or **Level 3**)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> Simulation time increases at higher levels, by as much as 30%.

- **Ignore Small Holes** – select **Auto Detect** (i.e., allow the software to detect and ignore small holes) or **Diameter smaller than** and enter a parameter in the field (i.e., specify a diameter and holes smaller than the diameter will be ignored)

8. Click the **Signal Nets** tab.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. From the **Signal Nets** tab, set the following:

    - **Error Tolerance** – select a tolerance level from the drop-down menu (i.e., **Default**, **0.02**, **0.04**, **0.06**, **0.08**, **0.1**, **0.2**, **0.5**, or **1.0**)
    - **Conductor Modeling** – select **Mesh Inside** or **Impedance Boundary** from the drop-down menu

      > **Important:**
      >
      > When using surface roughness, users must select **Impedance Boundary** from the **Conductor Modeling** drop-down menu.

    - **General Mode Only: Improved Metal Loss Handling** – check the box to activate.

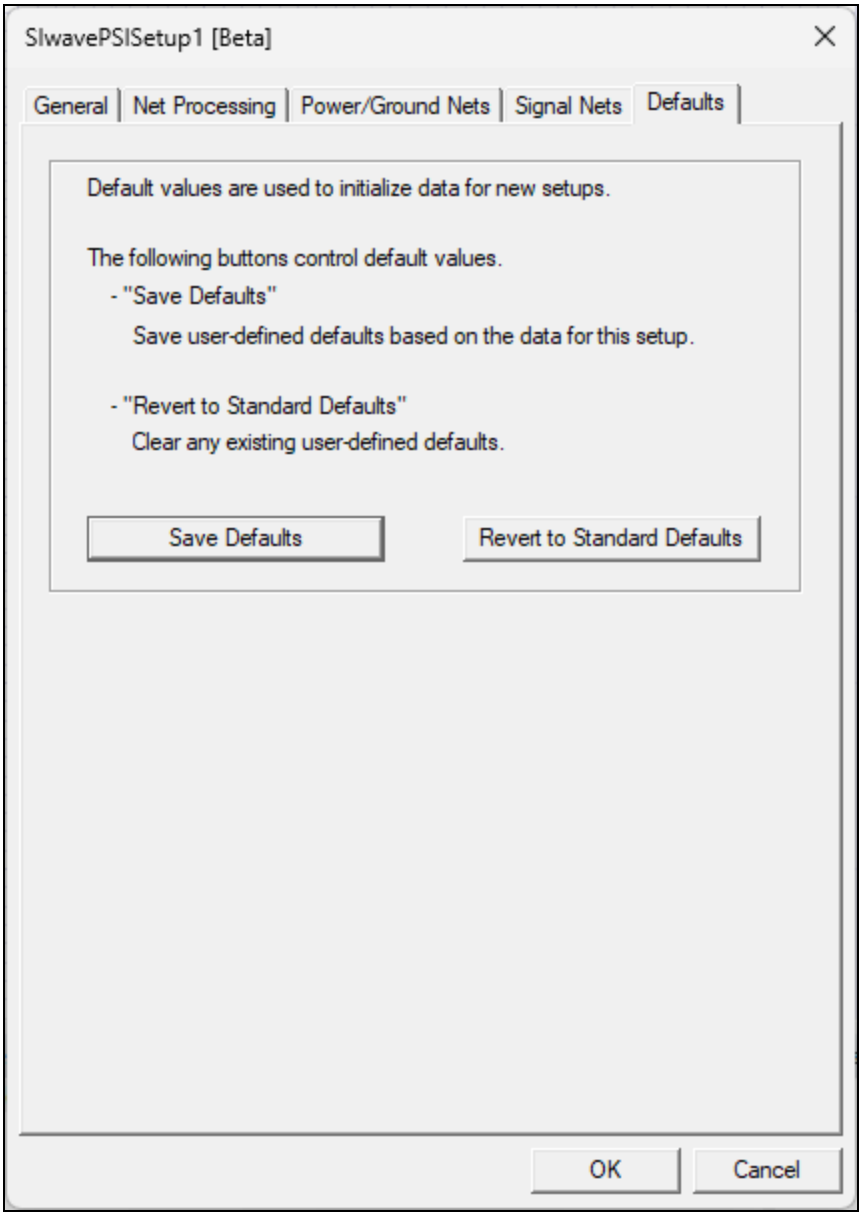
      > **Warning:**
      >
      > Enabling this option can significantly slow simulation time.

    - **Improved Dielectric Fill Refinement in Metal Layers** – check the box to activate or deactivate **Improved Dielectric Fill Refinement**

      > **Warning:**
      >
      > Activating **Improved Dielectric Fill Refinement in Metal Layers** can significantly slow simulation time.

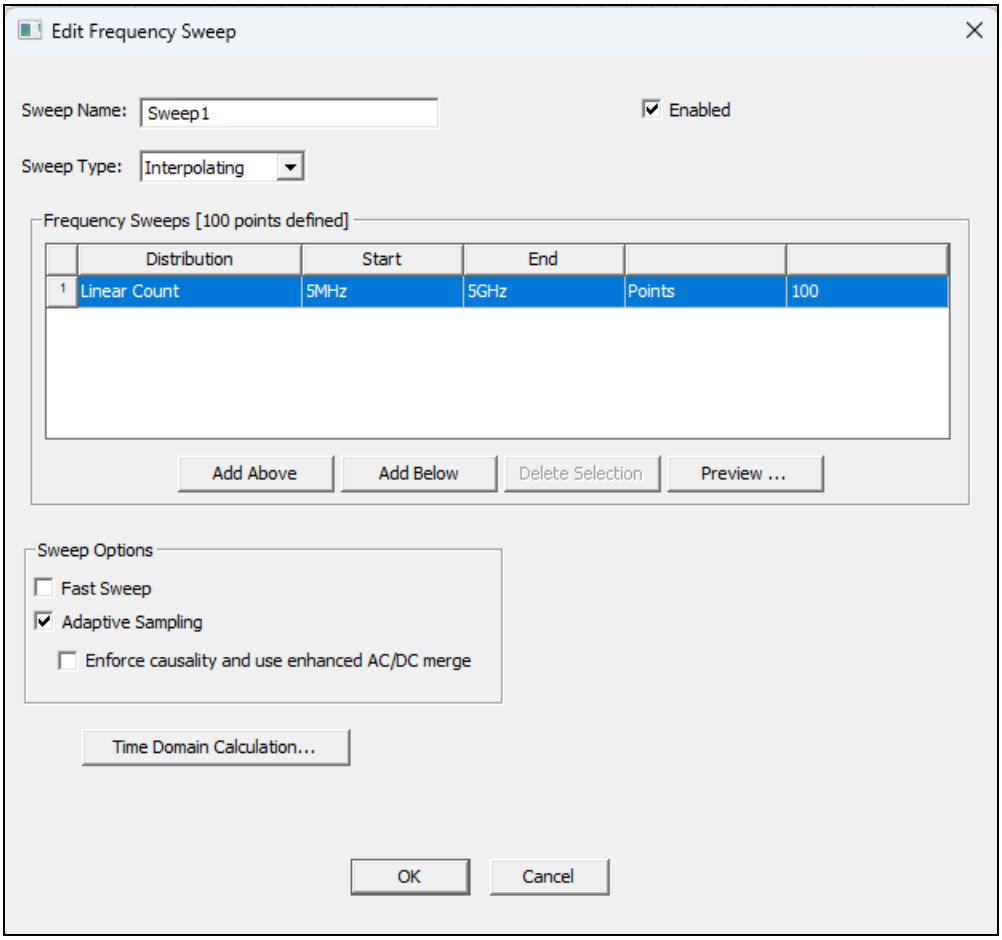
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Click the **Defaults** tab.



11. From the **Defaults** tab, either **Save Defaults** or **Revert to Standard Defaults**.
12. Click **OK** to add the SIwave-PSI solution setup to the active design and simultaneously open the **Edit Frequency Sweep** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



13. From the **Edit Frequency Sweep** window, click **OK** to accept the sweep with its default values or specify the following:

> **Note:**
>
> The available tabs and options change with the selected sweep type.

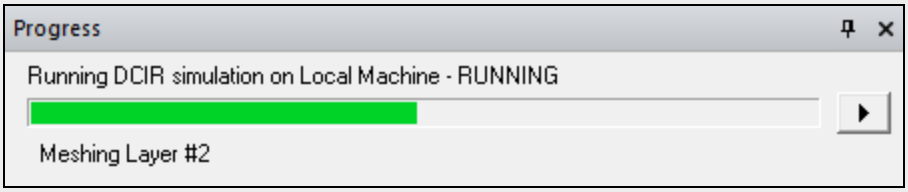
- **Sweep Name** field – enter a name for the sweep.
- **Sweep Type** drop-down menu – select **Discrete** or **Interpolating**:
  - **Discrete** – Generates field solutions at specific frequency points in a frequency range. Ideal when only a few frequency points are appropriate to accurately represent the results in a frequency range.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Interpolating** – Estimates a solution for an entire frequency range. Ideal when the frequency range is wide and the frequency response is smooth. When Interpolating is selected, the **Interpolation** tab appears.

  > **Note:**
  >
  > The information specified for an Interpolating sweep dictates the amount of information that can be viewed on a post-processing plot.

- **Enabled** – check to enable the sweep.
  - **Frequency Sweeps** area – use **Add Above**, **Add Below**, and **Delete** to add or remove sweeps. Each row in the table represents a separate sweep and each column is either a selectable parameter or a value field. You can add multiple frequency sweeps to the same setup. Specify the **Start** and **End** frequency for each sweep, and make selections from the additional options specific to the **Distribution** type.

    > **Note:**
    >
    > **IC Mode Only:** If the user enters **0MHz** in the **Start** value field, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform can use a DCIR solver within the SIwave-PSI simulation to calculate DC point.
    >
    > 
    >
    >
    > To activate this beta feature, navigate to the **Beta Options** window as described in Activating the SIwave-PSI Solution Beta Feature steps 1-2, then check the box adjacent to **DCIR solver support for SIwave-PSI in IC Mode**.

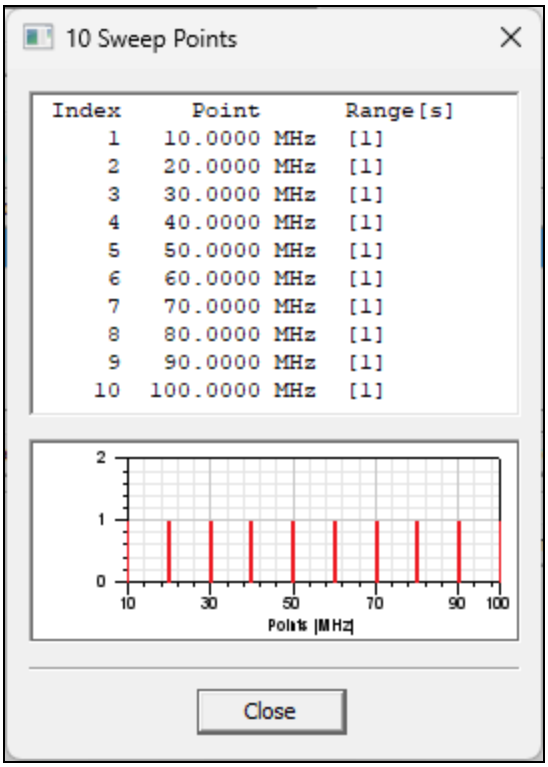
    - **Distribution** – select from the following:
      - **Linear Step** – A linear range of frequency points for which users specify a constant step size. The simulation will solve the frequency point at each step in the specified frequency range, including the start and stop frequencies (e.g. , specifying 10 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 **Start** frequency, 20 for the **Stop** frequency, and 2.5 for the **Step Size** instructs the simulation to compute a solution for frequencies of 10, 12.5, 15, 17.5, and 20).

- **Single Point** – Individual frequency points. Insert specific frequency points to solve in the frequency range after uniform frequency points. For **Discrete** sweep types only.
- **Single Point Sweep** – Adds a set of 10 Single Point Sweeps, defaulting from 1 GHz to 10 GHz in increments.
- **Linear Count** – A linear range of frequency points for which users specify the number (count) of points within the frequency range. The simulation divides the frequency range into the count specified and solve each frequency point in the count.
- **Log Scale** – A range of points for which users specify **Start**, **End**, and the number of **Samples**.The simulation assigns the sampled points using intervals based on a logarithmic scale.

- Click **Preview** to view a preview of the sweep, as currently defined.



- From the **Sweep Options** area, check boxes to activate any/all 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.

> **Note:**
>
> **Sweep Options** are only available if the **Sweep Type** is **Interpolating**.

- **Fast Sweep** check box – check to improve runtime by requiring fewer samples to simulate. Uses more memory.
- **Adaptive Sampling** check box – check to determine the number of sample frequency points from the response over the frequency band, adding more points around rapidly changing responses (e.g., resonance peaks). Checking the **Adaptive Sampling** box activates the **Enforce causality and use enhanced AC/DC merge** box.
    - **Enforce causality and use enhanced AC/DC merge** check box – check to enforce causality and generate S-parameters at the DC point. By combining the DC results with the frequency-swept AC results, accuracy is improved over the entire simulation bandwidth.

14. Click **OK** to close the **Edit Frequency Sweep** window and add any sweeps.
15. From the **Edit Frequency Sweep** window, click **OK** to accept the sweep with its default values or change the default values. Refer to Adding a Frequency Sweep for more information.
16. **IC Mode Only:** If applicable, apply a technology file to the design.

> **Note:**
>
> Encrypted technologies are only supported by HFSS, Q3D, and RaptorX simulations.

17. Run the analysis to solve the setup. Refer to Running Simulations.
18. To add a PI type:
    a. From the **Layout Editor**, select the port.
    b. From the **Properties** window, select the **EM Design** tab. Then click the **SIwave-PI Type** value field and select from the following:
        - Default (the initial condition of a new port)
        - Coaxial Open
        - Coaxial Shorted
        - Gap
        - Lumped

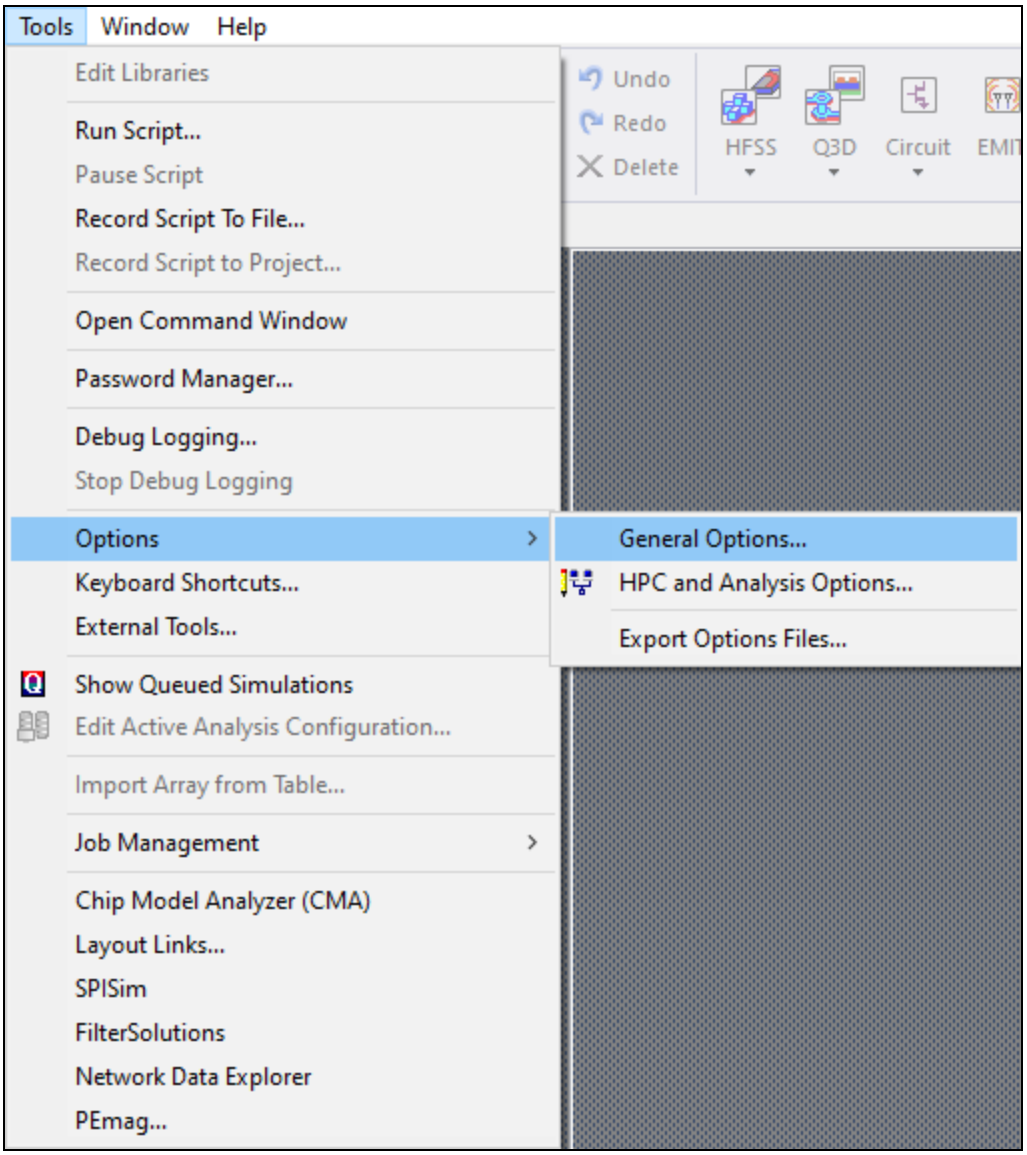
The procedure is complete.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Activating the SIwave-PSI Solution Beta Feature

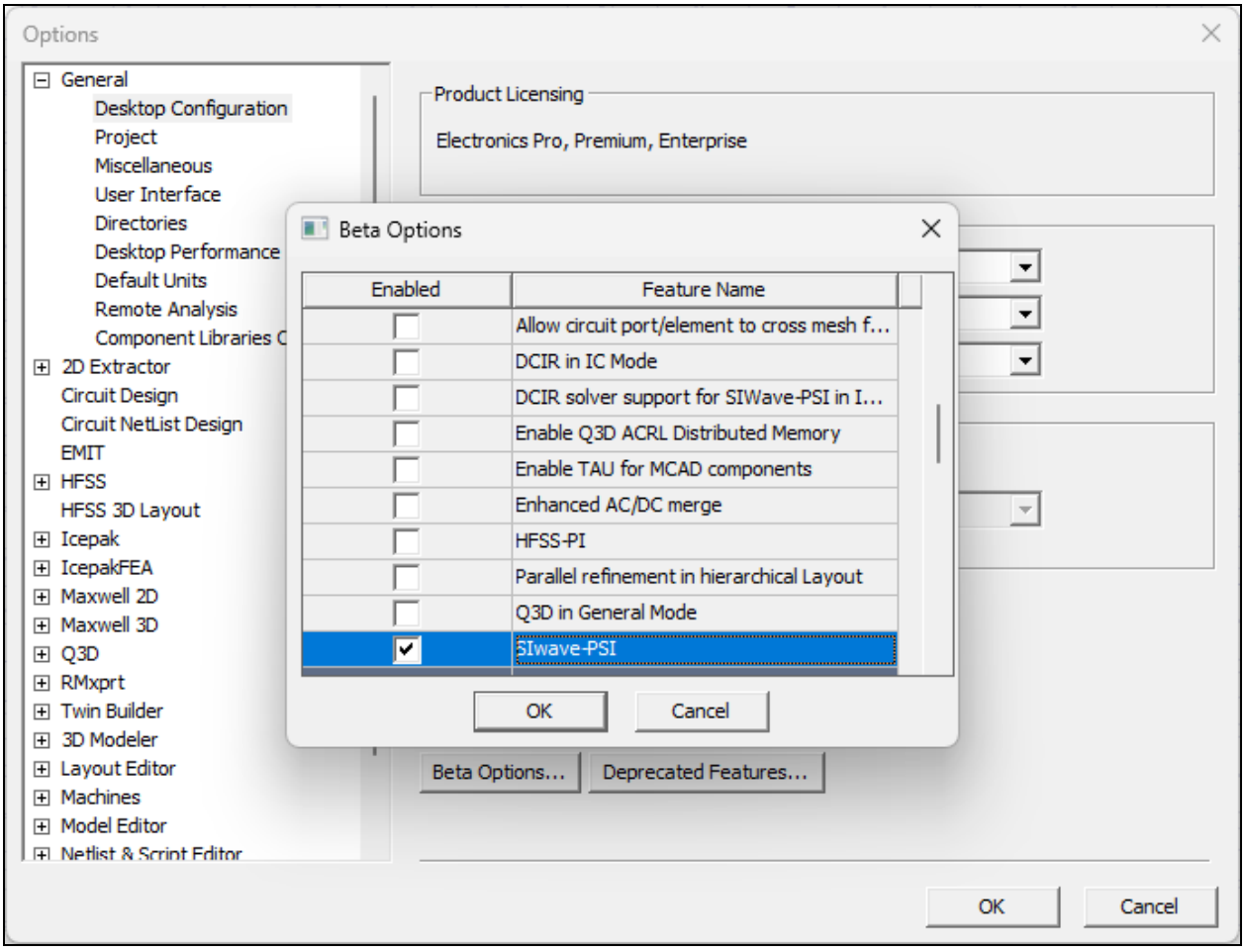
Follow these steps to enable the SIwave-PSI Solution Setup beta feature.

1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window.

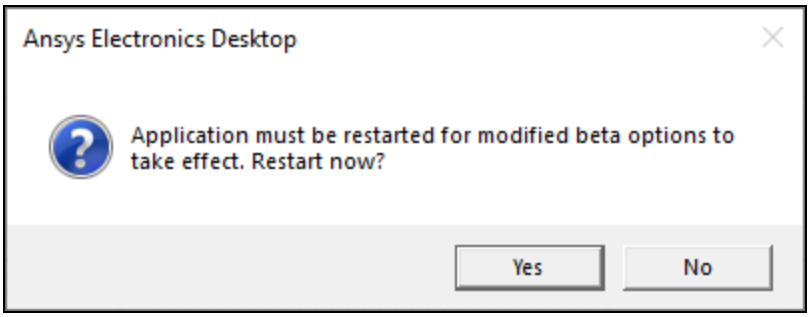


2. Expand **General** > **Desktop Configuration** and click **Beta Options** to open the **Beta Options** window.
3. Scroll down and check the **SIwave-PSI** box.

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** to close the **Beta Options** window.
5. Click **OK** to close the **Options** window and open a prompt to restart the software. Users may be asked to save their work.



6. Click **Yes** to restart immediately.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. After restarting the software, the SIwave-PSI solution setup is available from the locations described in the previous section.

The procedure is complete.

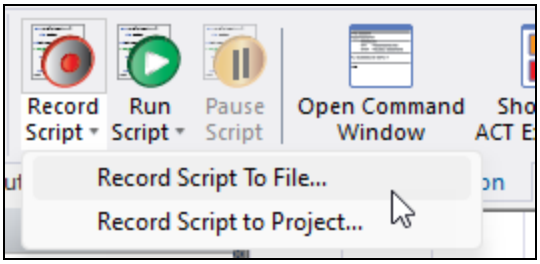
Using a Python Script to Configure an Ansys SIwave™ Solution Setup

Complete the steps in the following sections to start/stop recording a Python script (i.e., a *.**py** file), or to generate an Ansys ECAD database file (i.e., a ***.def** file) via Python script. Then import the newly created custom settings into an Ansys SIwave™ solution setup.

Recording a Python Script

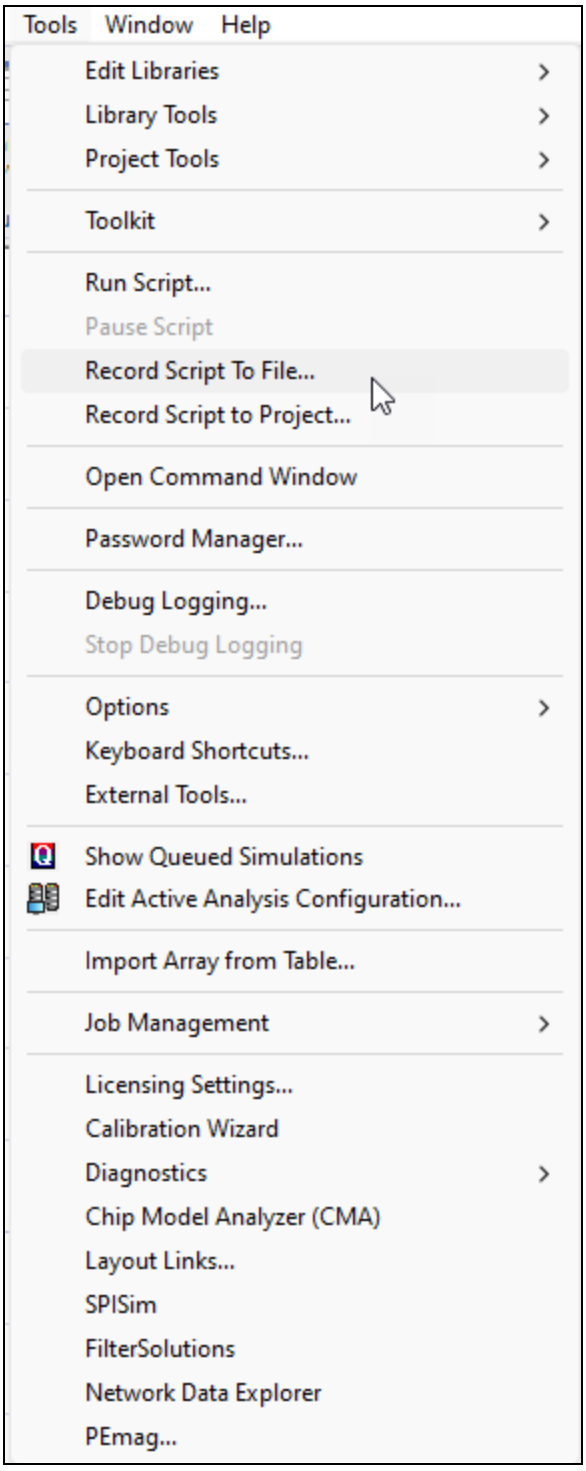
1. Do either of the following to open an explorer window:

- Navigate to the **Automation** tab and click **Record Script** > **Record Script To File**/**Record Script To Project**.



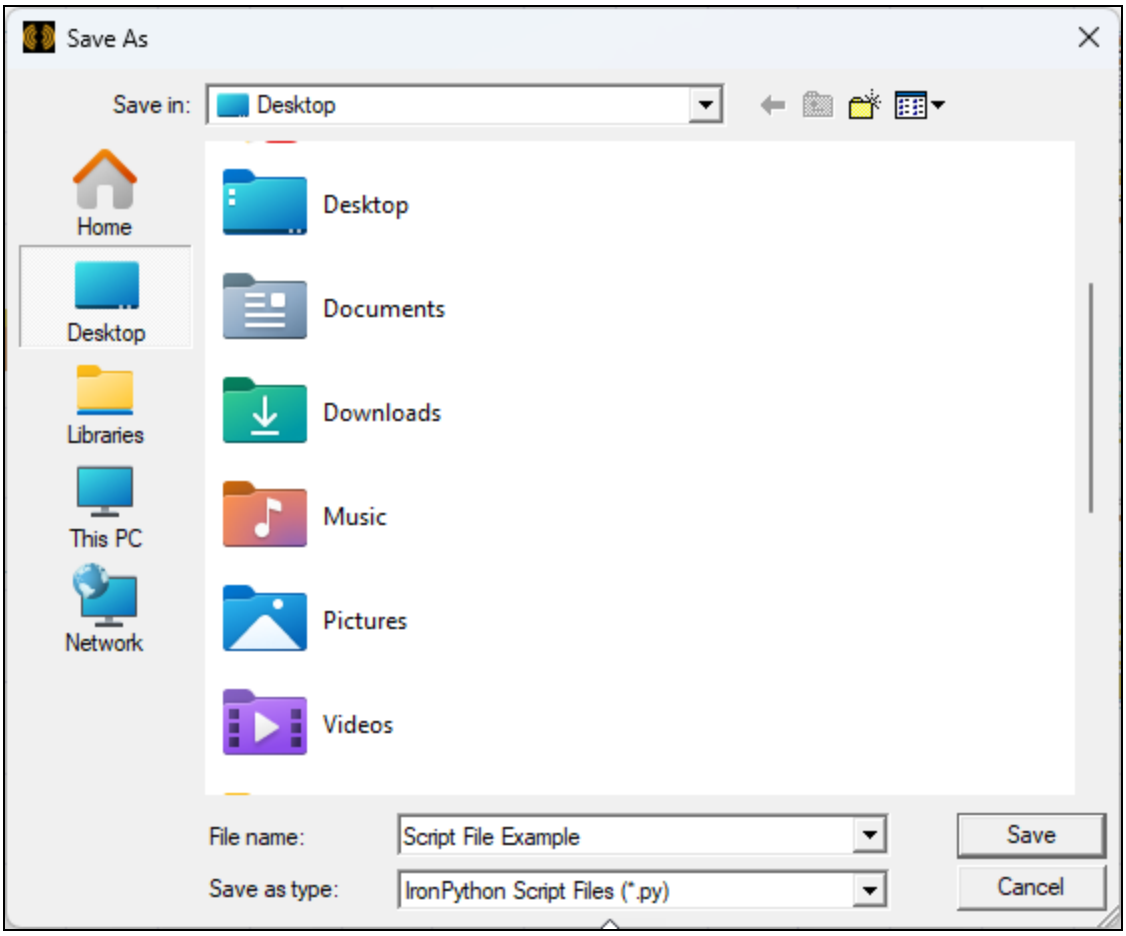
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From **Tools**, select **Record Script To File**/**Record Script To Project**.



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 an appropriate directory to save the Python file. Then enter a **File** name and click **Save**.

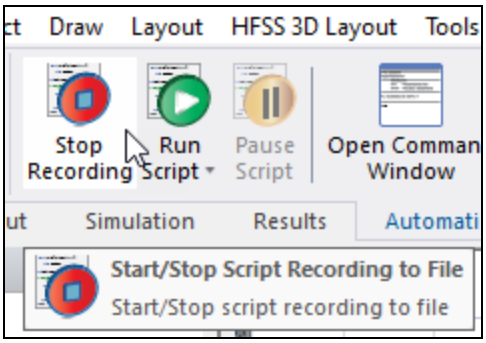


3. Complete any/all functions required. All actions taken within the Ansys Electronics Desktop (AEDT)™ electronics systems design platform that have a matching script are recorded and saved to the Python file.

4. To complete the Python and script and stop recording, do either of the following:

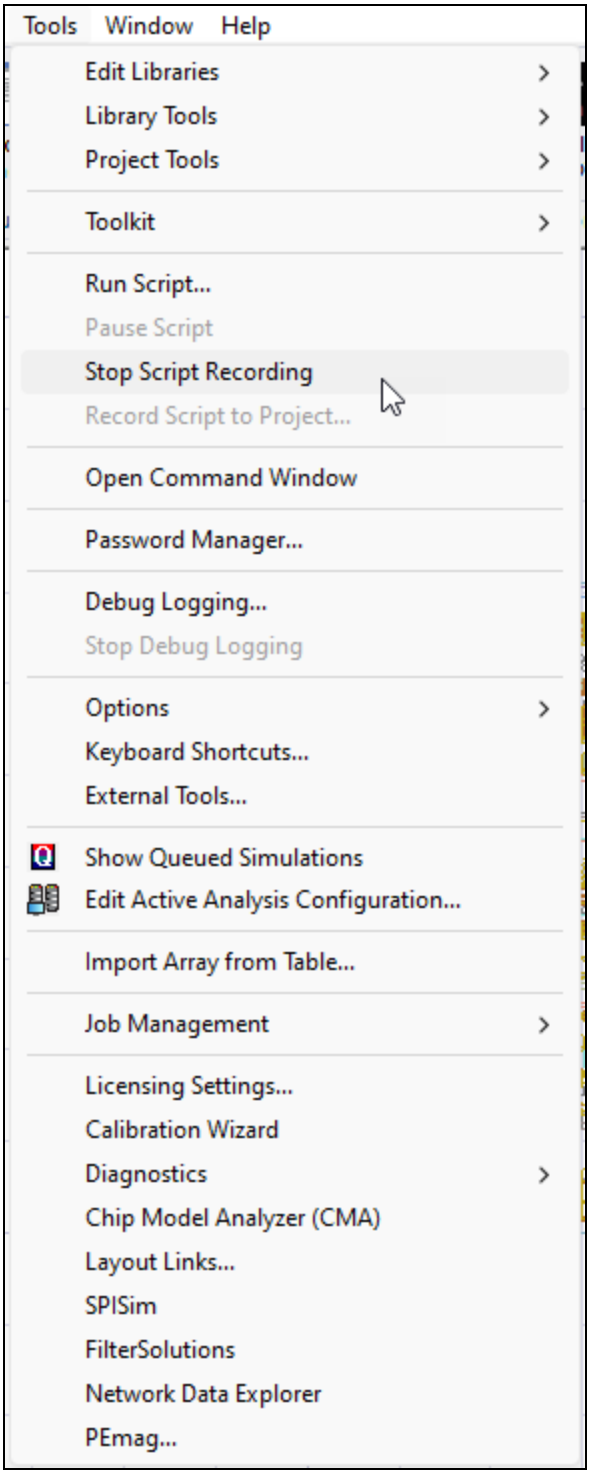
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Navigate to the **Automation** tab and click **Stop Recording**.



- From **Tools**, select **Stop Script Recording**.

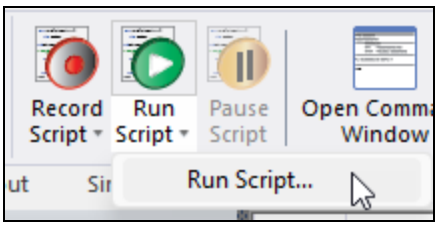
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Running a Python Script

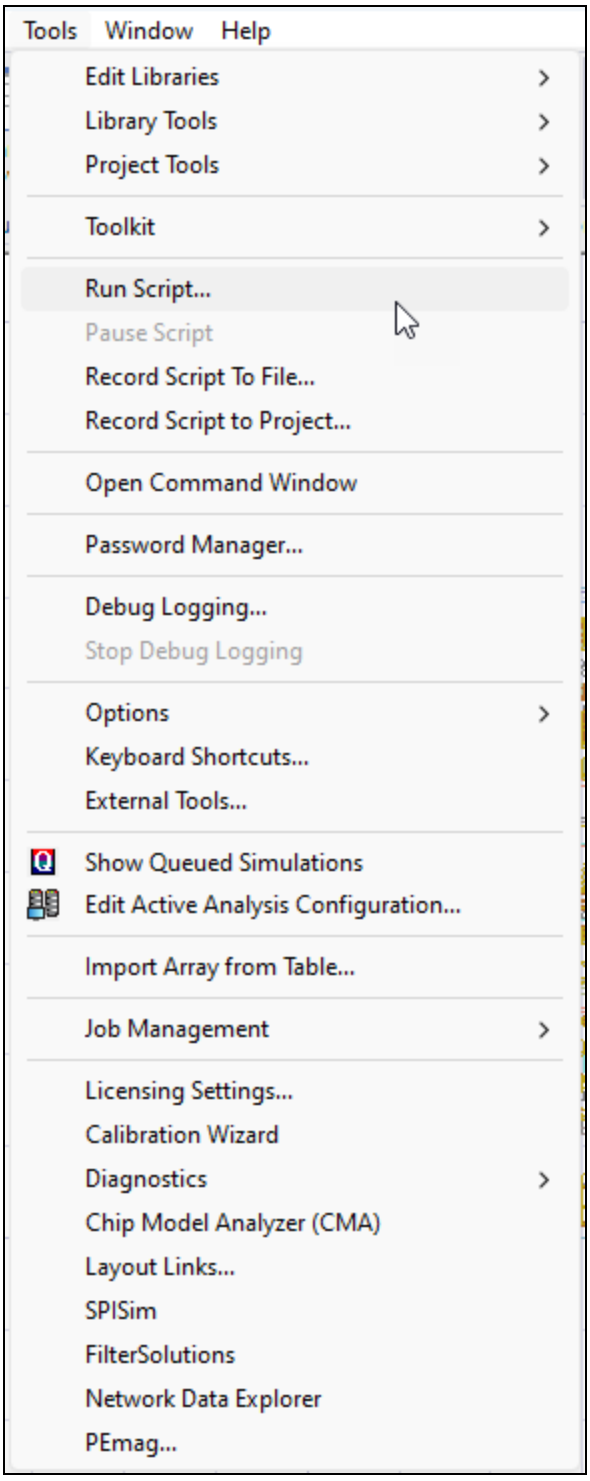
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Do either of the following:
    - Navigate to the **Automation** tab and click **Run Script** > **Run Script**.



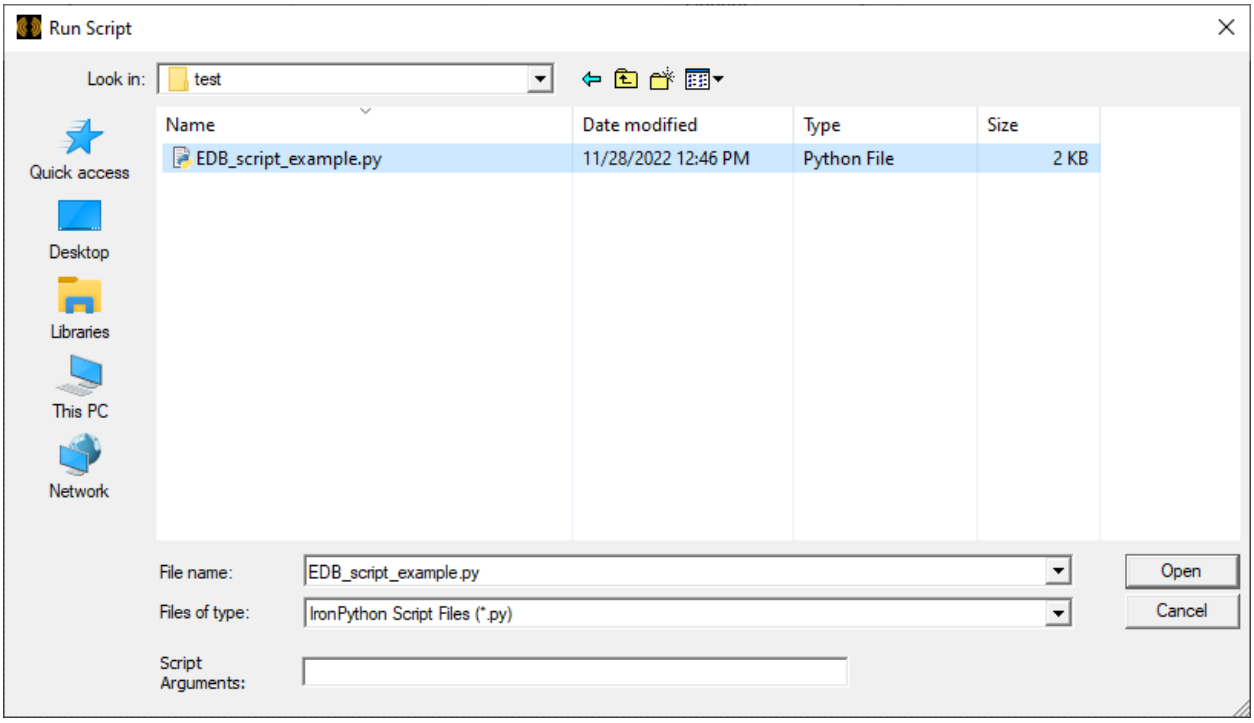
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From **Tools**, select **Run Script**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

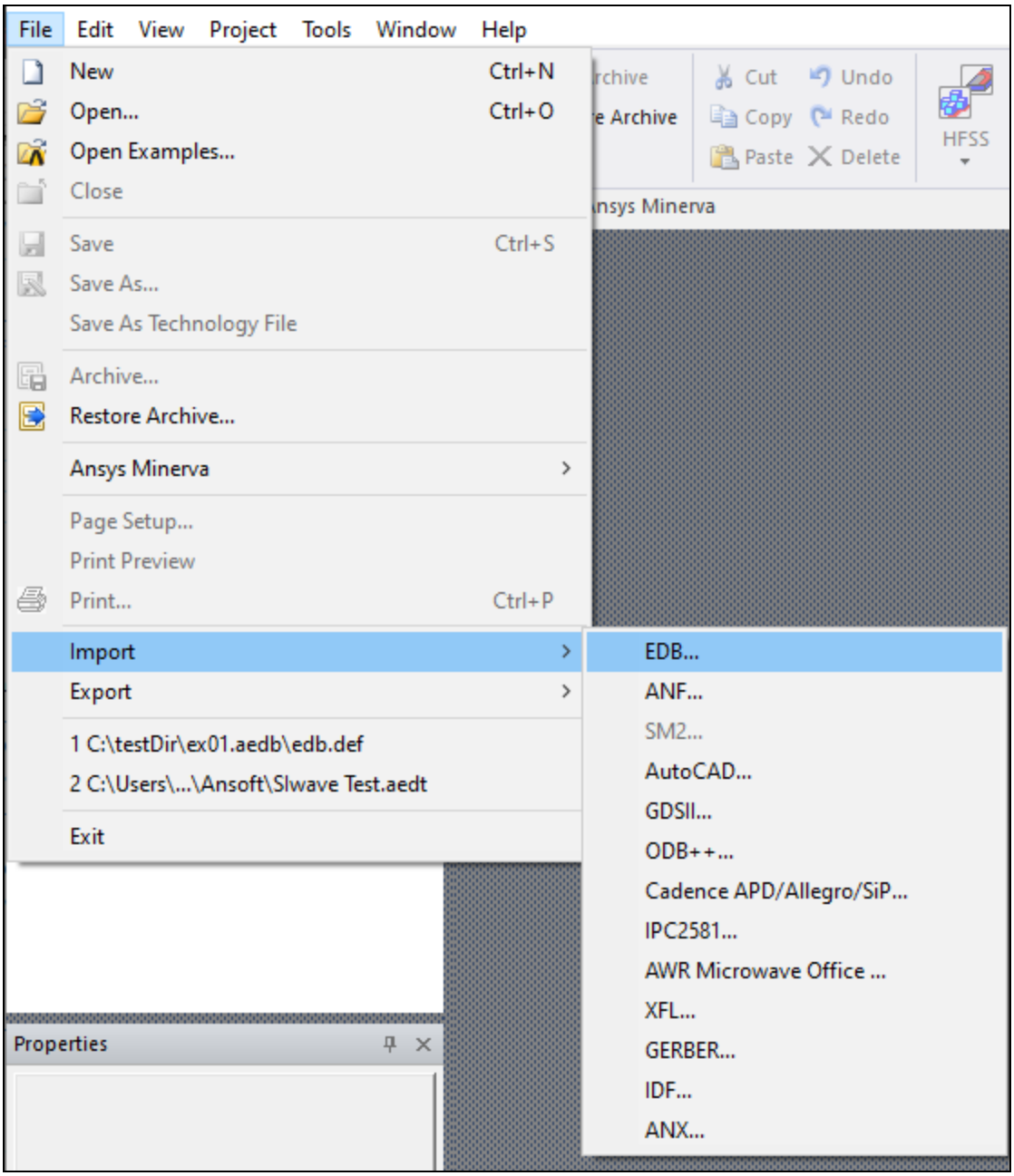
2. Navigate to the selected directory and select the **IronPython Script File** (e.g., **EDB_script_example.py**).



3. Click **Open**. The script generates a new Ansys ECAD database file at the location specified in the script (e.g., `temporaryDirectory = "C:\\testDir"`).

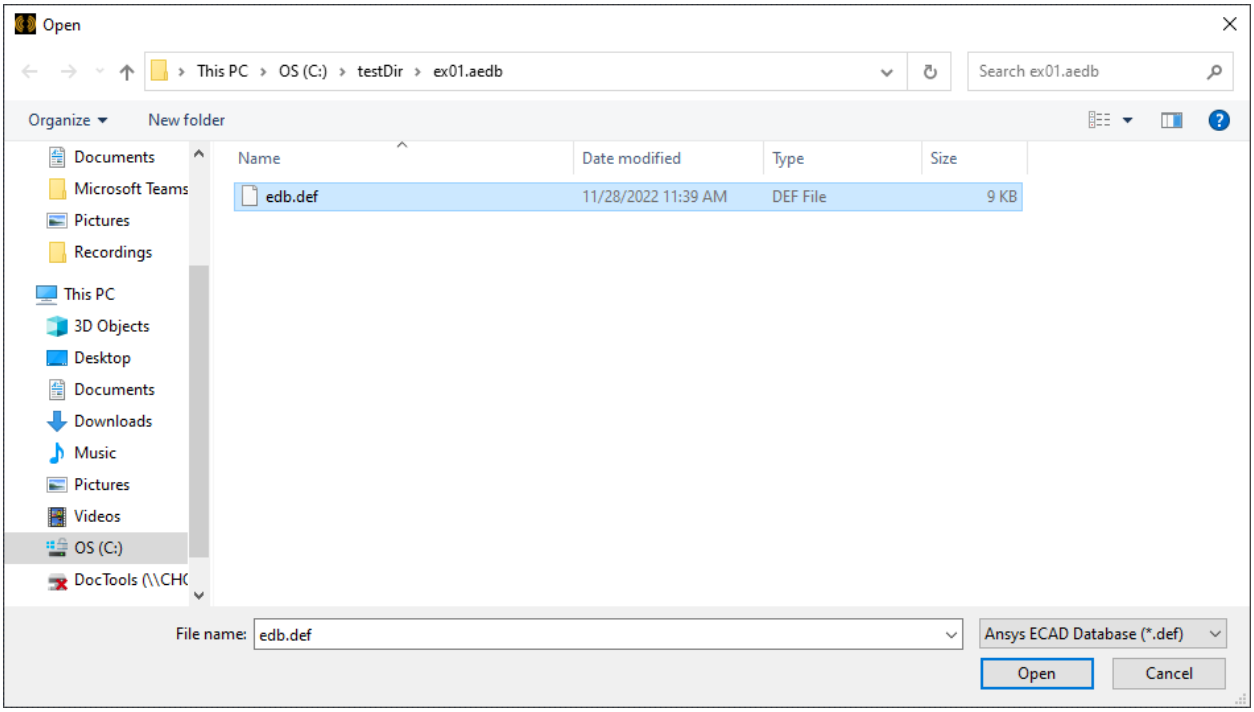
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. From **File**, select **Import** > **EDB** to open an explorer window.

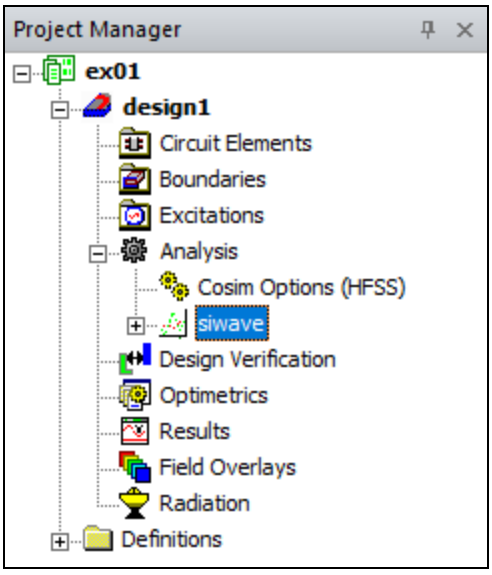


5. Navigate to the directory specified in the script file. Then select the Ansys ECAD database file (i.e., **edb.def**) and click **Open**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

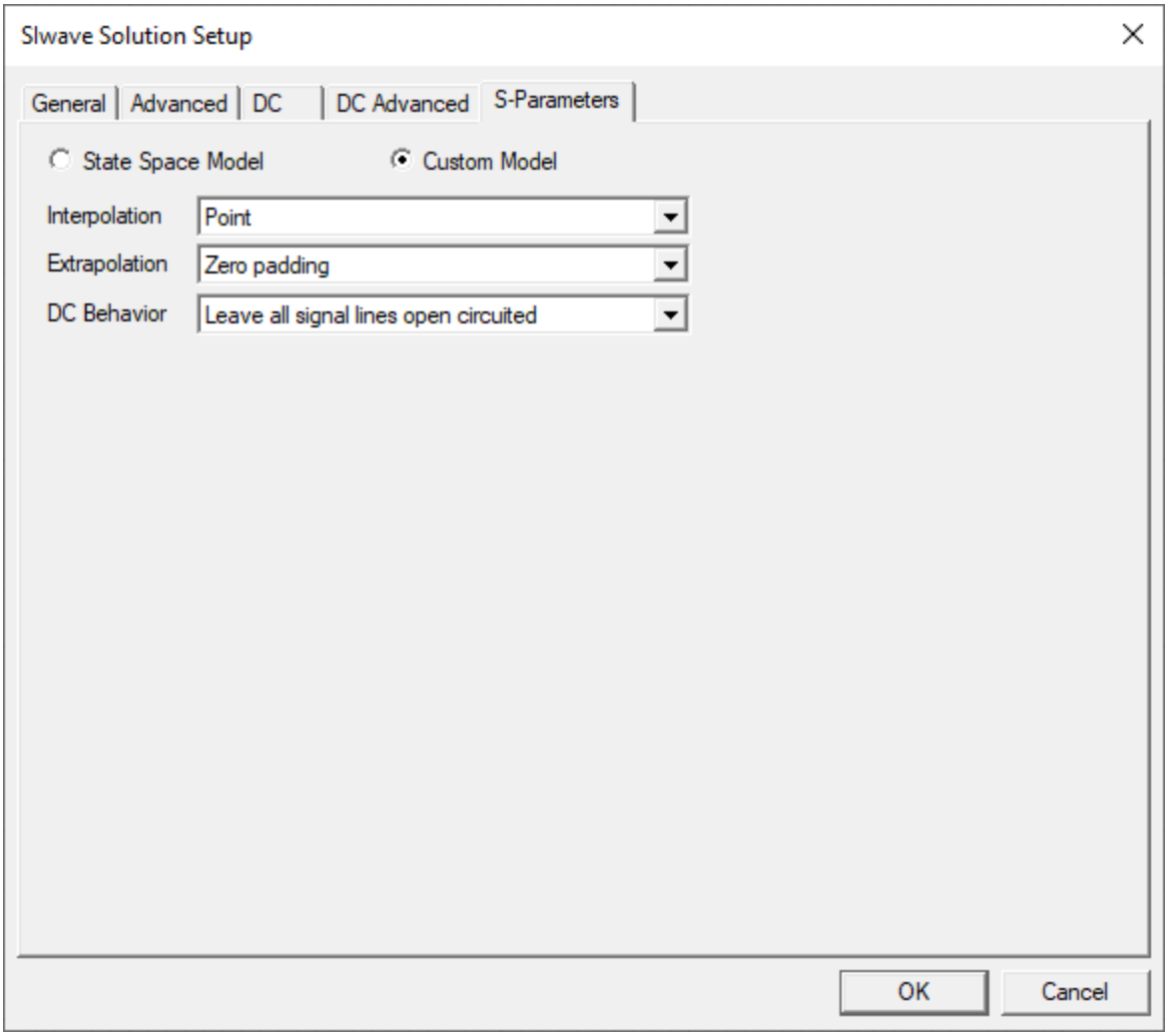


6. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** and double-click the Ansys SIwave™ setup (e.g., **siwave**) to open the **SIwave Solution Setup** window.



7. Navigate to the **S-Parameters tab** to view the newly imported custom model settings.

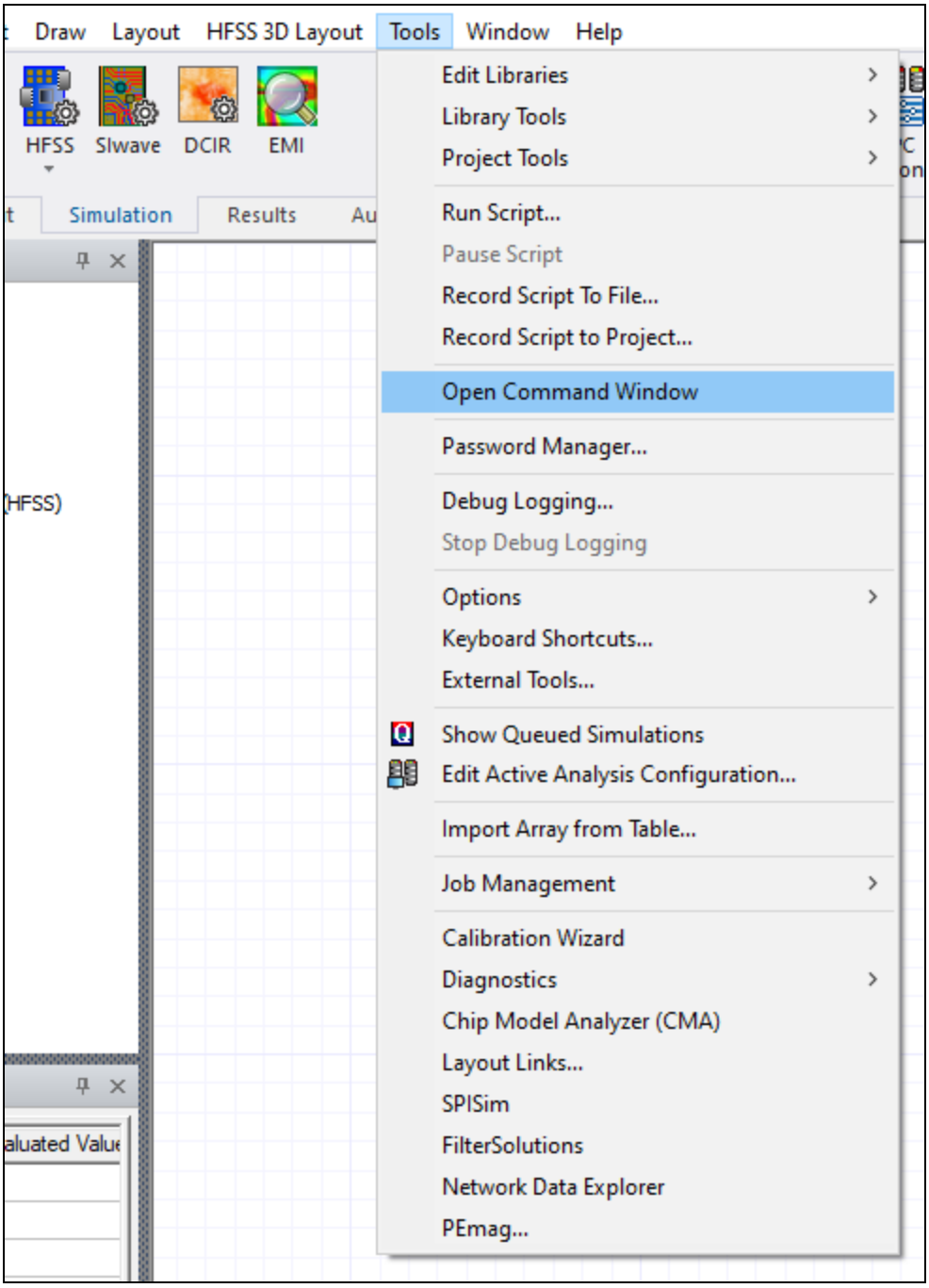
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Copying and Pasting Script Commands

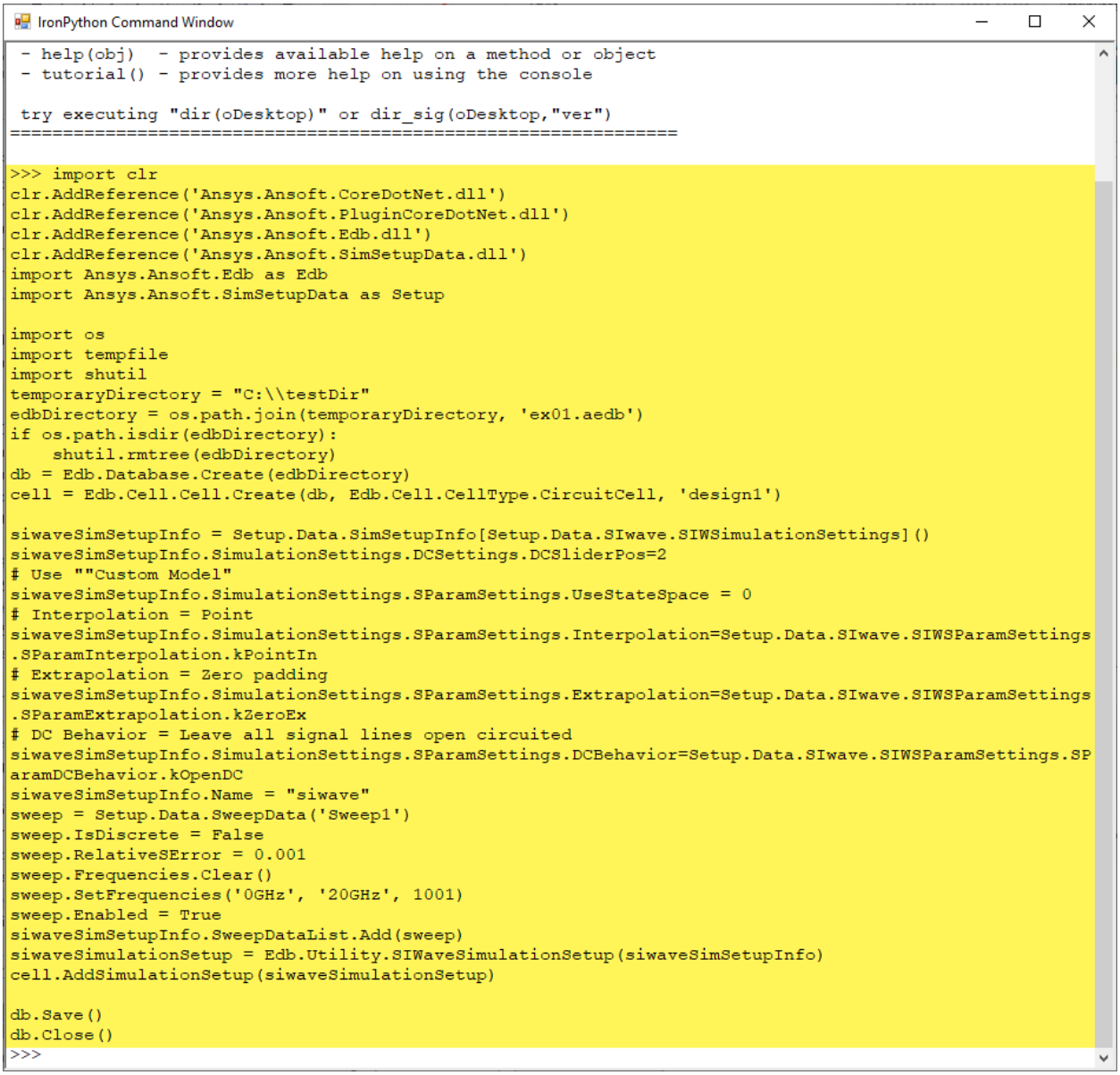
1. From **Tools**, select **Open Command Window** to open the **Iron Python Command Window**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Copy and paste (e.g., **Ctrl**+**C**, **Ctrl**+**V**) the script text at the command line. Then press **Enter**.

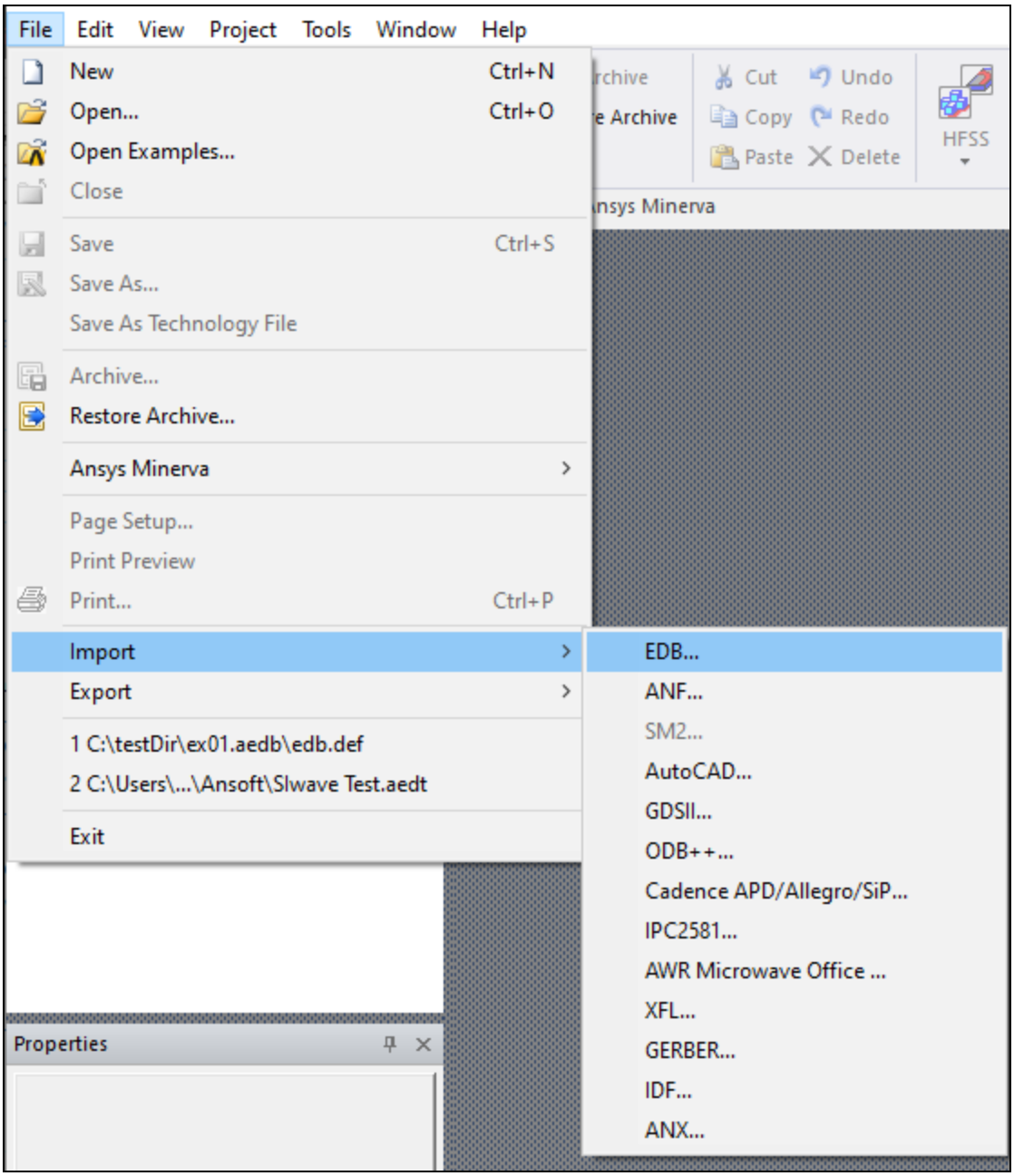
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. The script commands generate a new Ansys ECAD database file at the location specified in the script (e.g., **`temporaryDirectory = "C:\\testDir"`**).

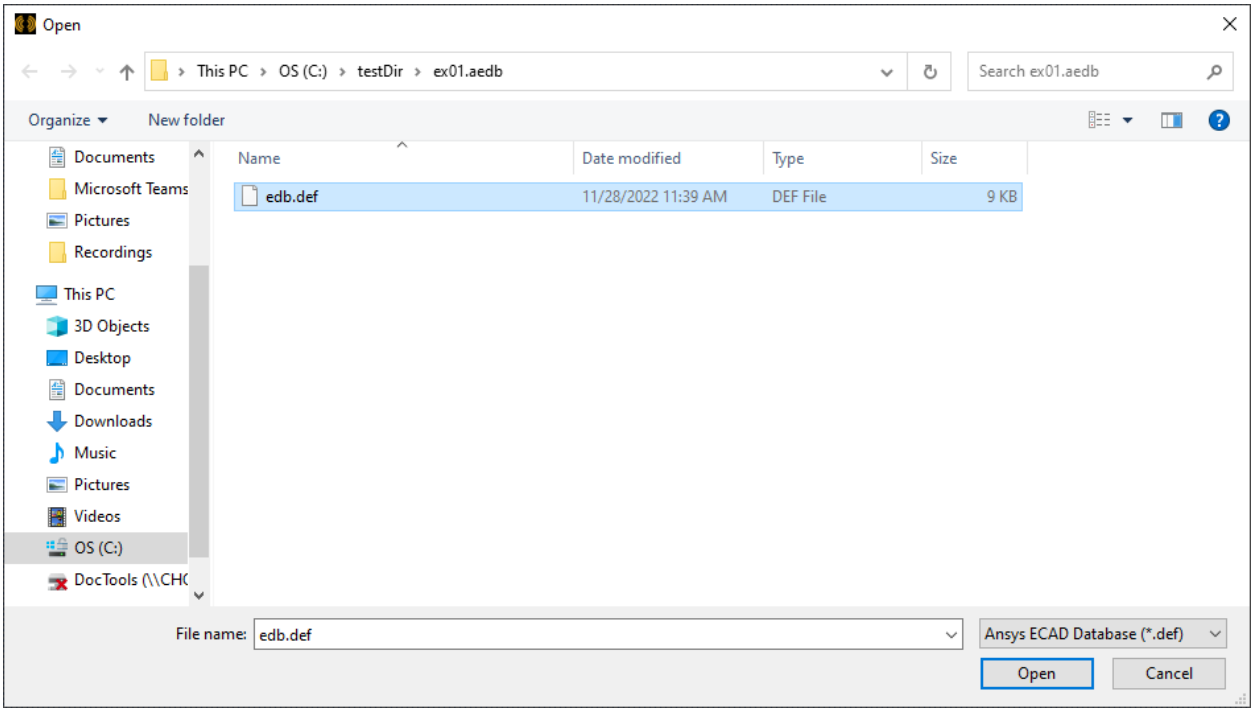
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. From **File**, select **Import** > **EDB** to open an explorer window.

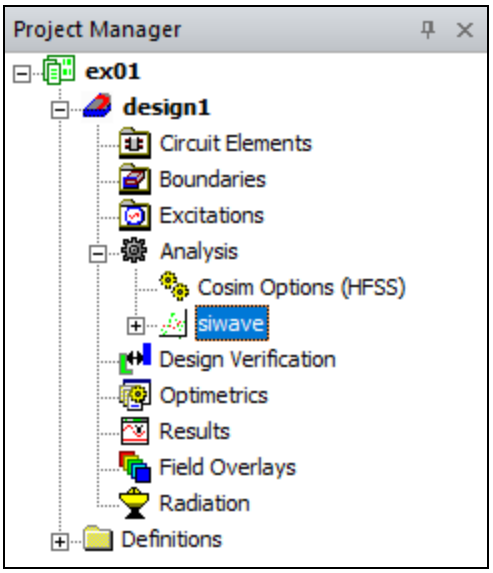


5. Navigate to the directory specified in the script text. Then select the Ansys ECAD database file (i.e., **edb.def**) and click **Open**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

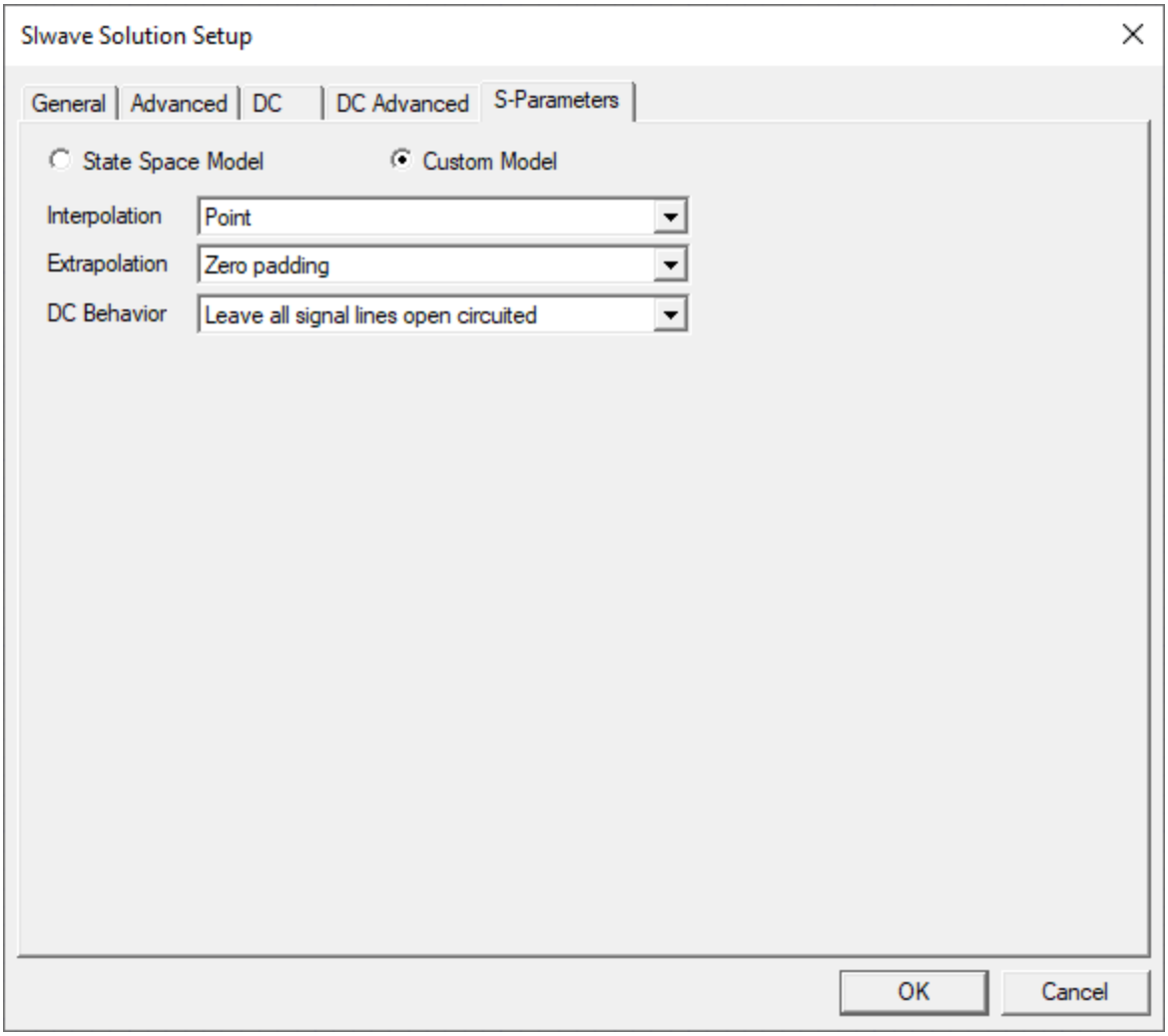


6. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** and double-click **siwave** to open the **SIwave Solution Setup** window.



7. Navigate to the **S-Parameters tab** to view the newly imported custom model settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



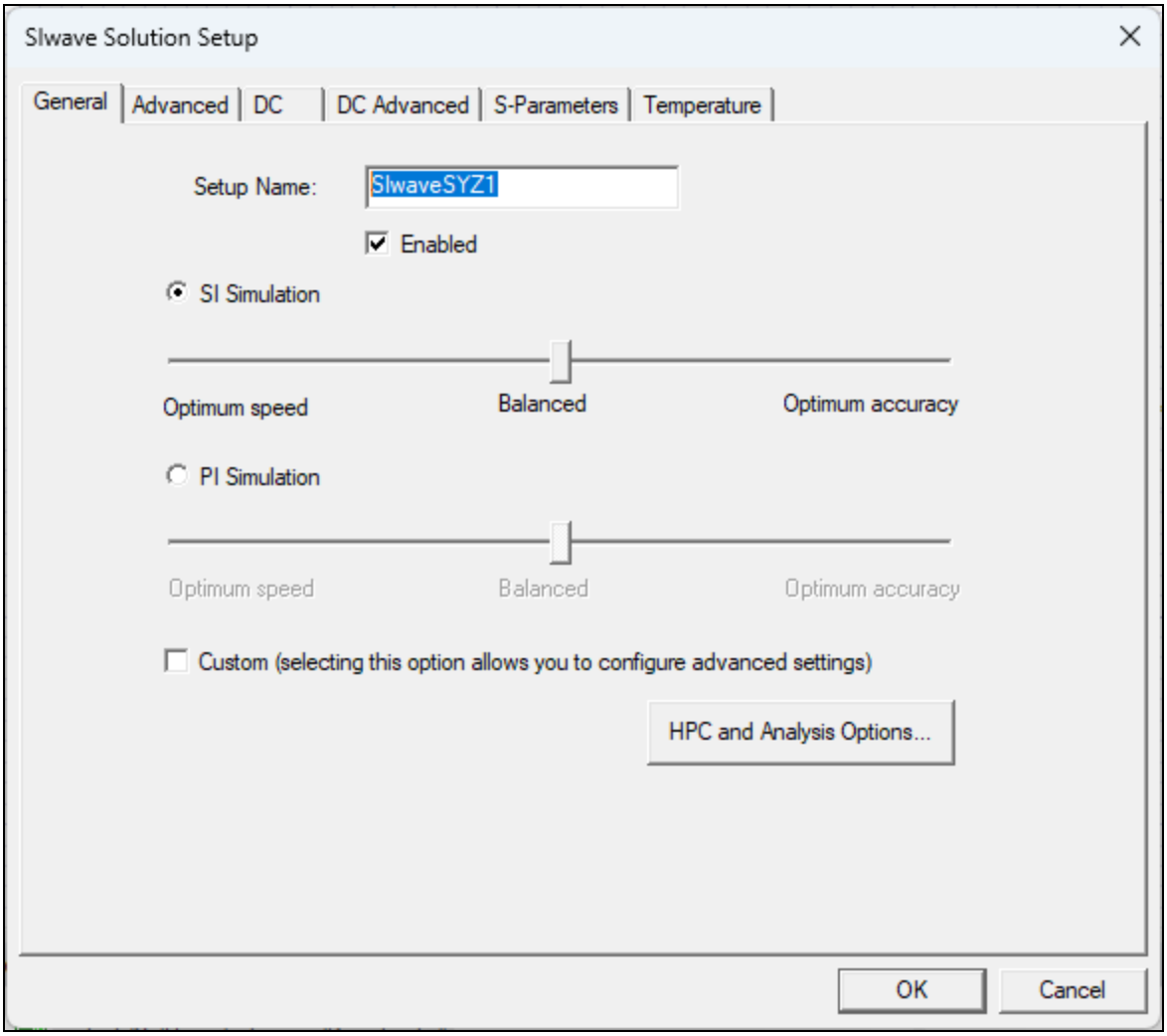
Selecting SIwave Solution Setup Options

Make further changes to Ansys SIwave™ simulations by modifying the general Ansys SIwave™ solution setup parameters.

General Tab

> **Note:**
>
> Parameters on this tab only affect SYZ and Near Field simulations.

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 **General** tab, do the following:

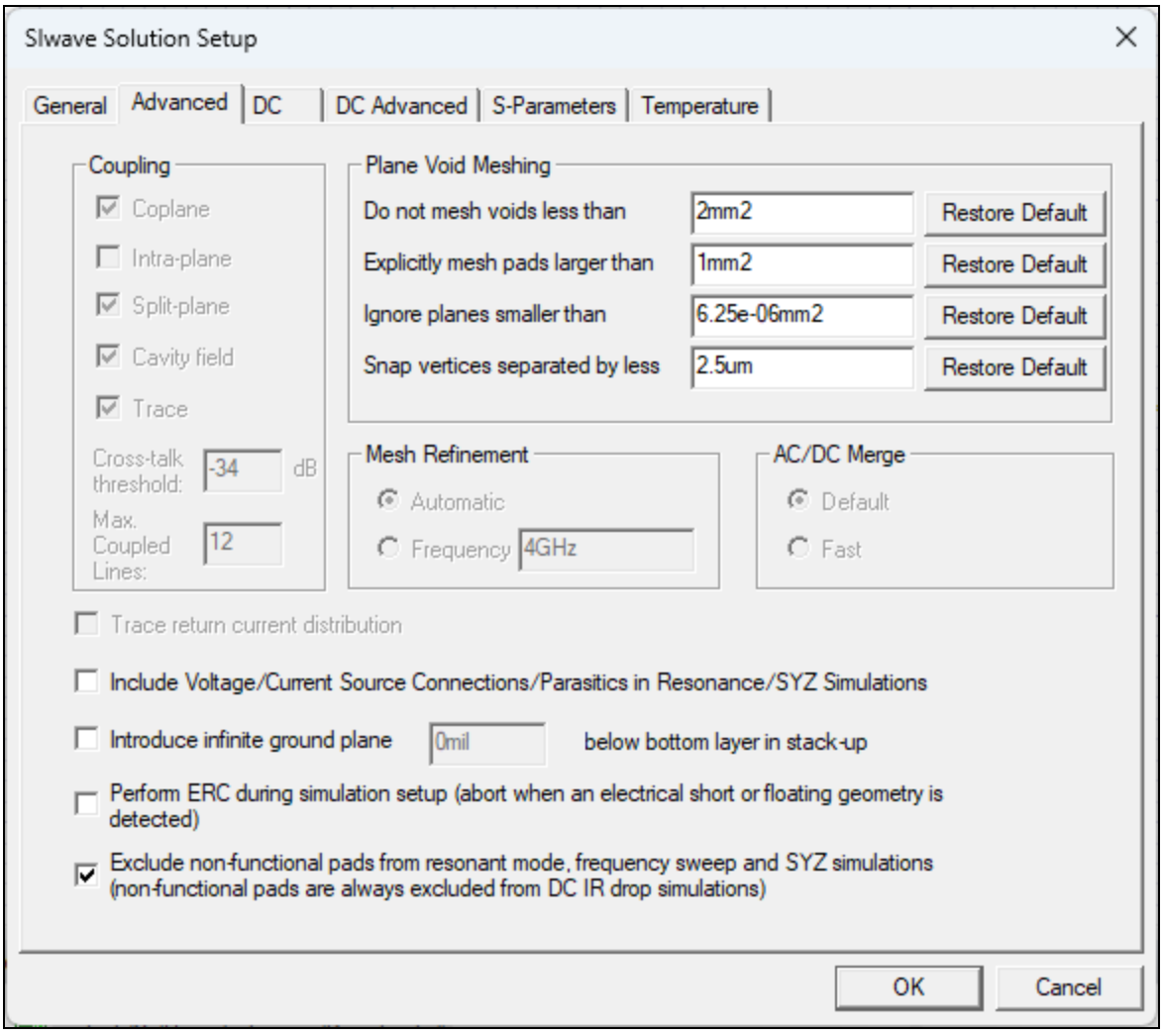
- Select either **SI simulation** (i.e., Signal Integrity) or **PI simulation** (i.e., Power Integrity), and move the corresponding slider to adjust speed and accuracy.
- For SI simulations:
  - **Optimum speed** — of the coupling options in the **Advanced** tab, only **Trace Coupling** is selected.
  - **Balanced** — of the coupling options in the **Advanced** tab, **Trace**, **Coplane**, **Split-Plane** and **Cavity Field Coupling** are selected.
  - **Optimum accuracy** — **Trace Return Current Distribution** is selected, in addition to **Balanced**.
- For PI simulations:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Optimum speed** — no coupling options selected.
  - **Balanced** — **Cavity Field Coupling** is selected.
  - **Optimum accuracy** — **Trace**, **Coplane**, **Split-Plane** and **Cavity Field** are selected.
- Check the **Custom** box allows to configure additional options in the **Advanced** tab.
- Click **HPC and Analysis Options** to open the HPC and Analysis Options window.

Advanced Tab

> **Note:**
>
> Parameters on this tab only affect SYZ and Near Field simulations.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

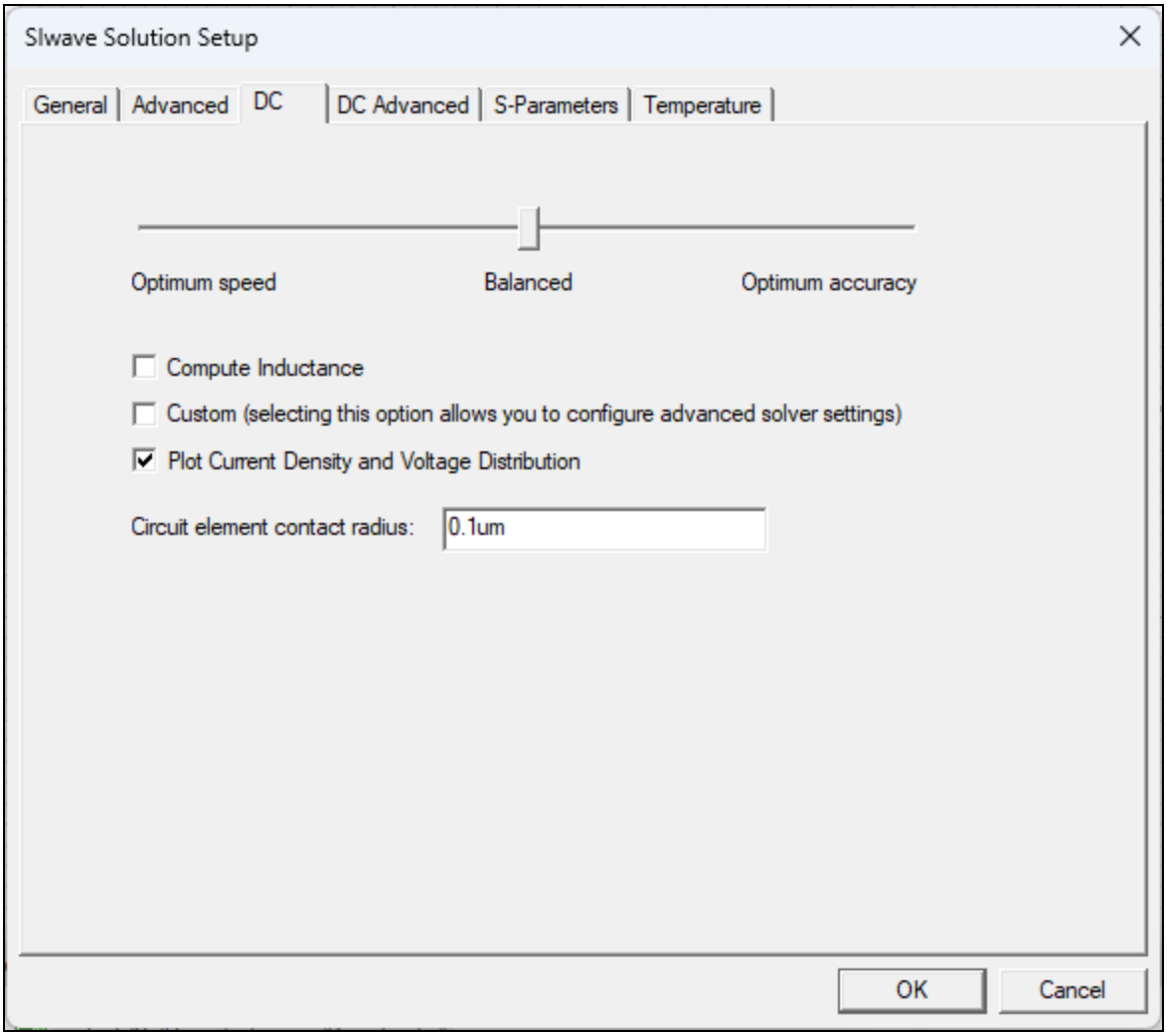
From the **Advanced** tab, do the following:

- **Coupling** — available only if the **Custom** box is checked in the **General** tab. If the **Custom** box is checked in the **General** tab, select any/all of these options.
- **Plane Void Meshing** — specify plane void meshing, control geometry defeaturing, and vertex snapping measurements and units, or **Restore Default** settings.
- **Mesh Refinement** — select **Automatic** or specify a **Frequency**.
- **Trace return current distribution** — available only if the **Custom** box is checked in the **General** tab. Check this option to accurately model the change of the characteristic impedance of transmission lines caused by a discontinuous ground plane. Instead of injecting the return current of a trace into a single point on the ground plane, the return current for a high impedance trace is spread out. The trace return current is not distributed when all traces attached to a node have a characteristic impedance of less than 75 ohms, or if the difference between two connected traces is less than 25 ohms.
- **Include Voltage/Current Source Connections/Parasitics in Resonance/SYZ Simulations** — determine whether to include the effect of parasitic elements from voltage and current sources.
- **Introduce infinite ground plane** — specify the elevation of the infinite unconnected ground plane to be placed under the design. The ground plane serves as a voltage reference for traces and planes if they are not defined in the layout.
- **Perform ERC during simulation setup** — determine whether to perform error checking while generating the solver input file. In some designs, the same net may be divided into multiple nets with separate names; these nets are connected at a “star point.” To simulate these nets, error checking for DC shorts must be deactivated. All overlapping nets are internally united during simulation.
- **Exclude non-functional pads from resonant mode, frequency sweep and SYZ simulations** — determine whether to remove non-functional pads. All layers in which a via does not make an electrical connection to any other geometry do not contain a pad. By default, non-functional pads are not included in the simulation. Typically non-functional pads are removed when a PCB is manufactured.

DC Tab

> **Note:**
>
> Parameters on this tab only affect DC IR simulations. However, if **Compute exact DC point** is checked on the **Edit Frequency Sweep** window during SYZ solution setup, then a DCIR simulation is effectively run during SYZ simulation.

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 **DC** tab, do the following:

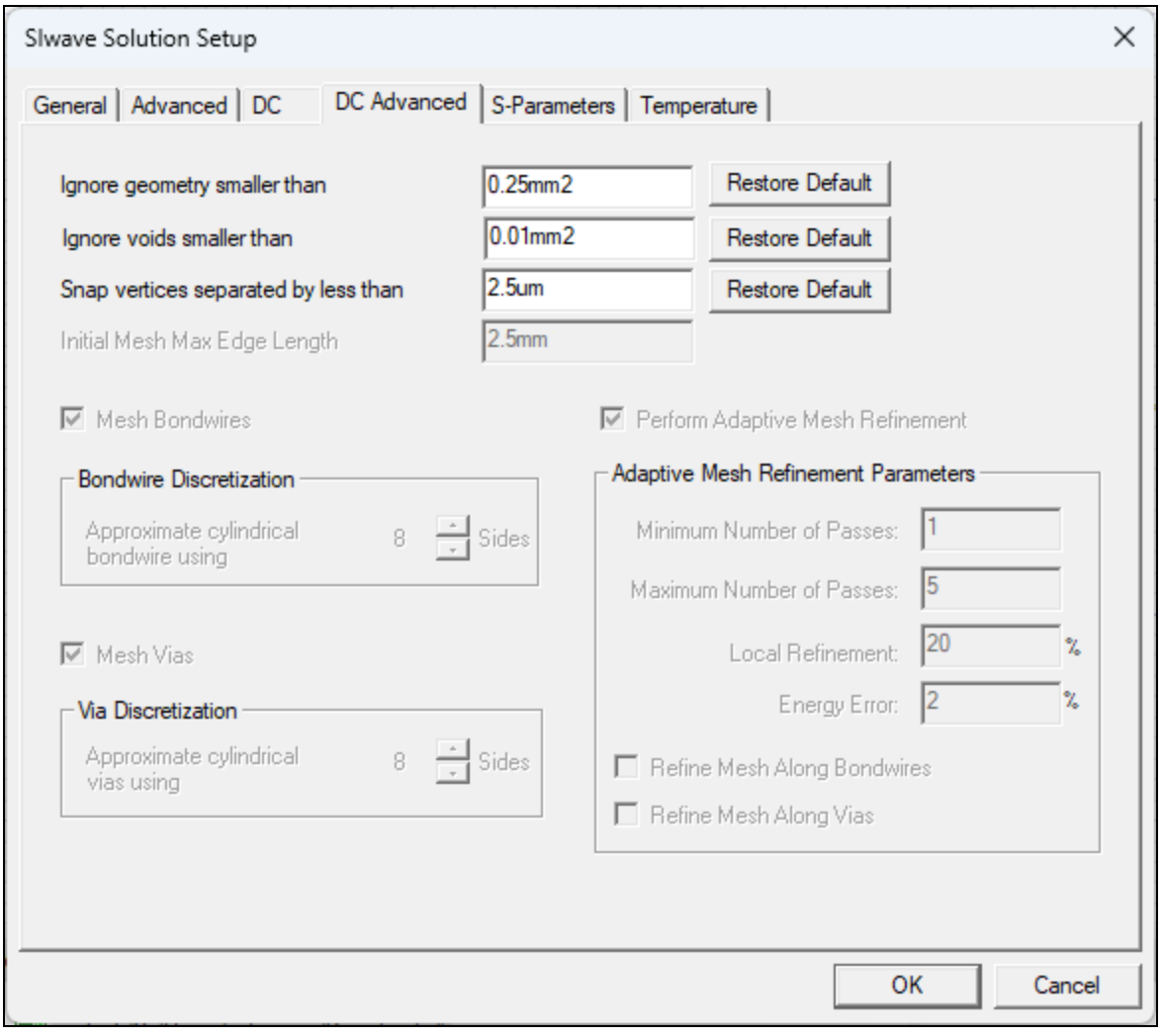
- Use the slider to balance speed and accuracy:
  - **Optimum Speed** — all options in the **DC Advanced** tab are deactivated.
  - **Balanced** — the **Mesh Bondwires**, **Mesh Vias** and **Automatic Mesh Refinement** are selected.
  - **Optimum Accuracy** — the mesh are refined on bondwire and vias, and more aggressive refinement parameters are specified.
- **Compute Inductance** — check to compute inductance by integrating the vector potential at DC using an efficient, multi-level fast multipole method with two-level parallelization. In some cases, this may require significant computational time and memory.
- **Custom** — check to use simulation settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Plot Current Density and Voltage Distribution** — uncheck to save disk space if post-processing voltage and current plots is inappropriate (e.g., if users export a SPICE netlist after a DC simulation, it may not be helpful to save any field data to disk).
- **Circuit Element Contact Radius** — enter a value. Equal potential is enforced in the contact radius, and no mesh refinement is done inside the contact region.

DC Advanced Tab

> **Note:**
>
> Parameters on this tab only affect DC IR simulations. However, if **Compute exact DC point** is checked on the **Edit Frequency Sweep** window during SYZ solution setup, then a DCIR simulation is effectively run during SYZ simulation.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DC Advanced options include the following:

- **Ignore geometry smaller than** — geometry whose area (after being combined with any overlapping geometry) falls under this threshold is ignored during simulation.
- **Ignore voids smaller than** — geometry whose area (after being combined with any overlapping geometry) falls under this threshold is ignored during simulation.
- **Snap vertices separated by less than** — entities (e.g., vias) and vertices that are separated by less than this specified distance may be snapped together during solver pre-processing.
- **Initial Mesh Max. Edge Length** — specifies the maximum edge length selected in the mesh (before any adaptive refinement occurs).
- **Mesh Bondwires** — check if users do not want to model bondwires as point connections to mesh. Specify the **Bondwire Discretization** (the number of sides used to approximate cylindrical bondwires). Specify any value between 6 and 64. The default value is 8.
- **Mesh Vias** — check if users do not want to model vias as point connections to mesh. Specify the **Via Discretization** (the number of sides used to approximate cylindrical vias). Specify any value between 6 and 64. The default value is 8.
- **Perform Adaptive Mesh Refinement** — check to perform Adaptive Mesh Refinement, and populate the fields:
  - **Minimum Number of Passes** — an adaptive analysis does not stop unless the minimum number of passes specified has been completed, even if convergence criteria have been met.
  - **Maximum Number of Passes** — this value is the maximum number of mesh refinement cycles users want the simulation to perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops; otherwise, the adaptive analysis continues unless the convergence criteria are reached.

    > **Note:**
    >
    > The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform requesting more memory than is available or taking excessive time to compute solutions.

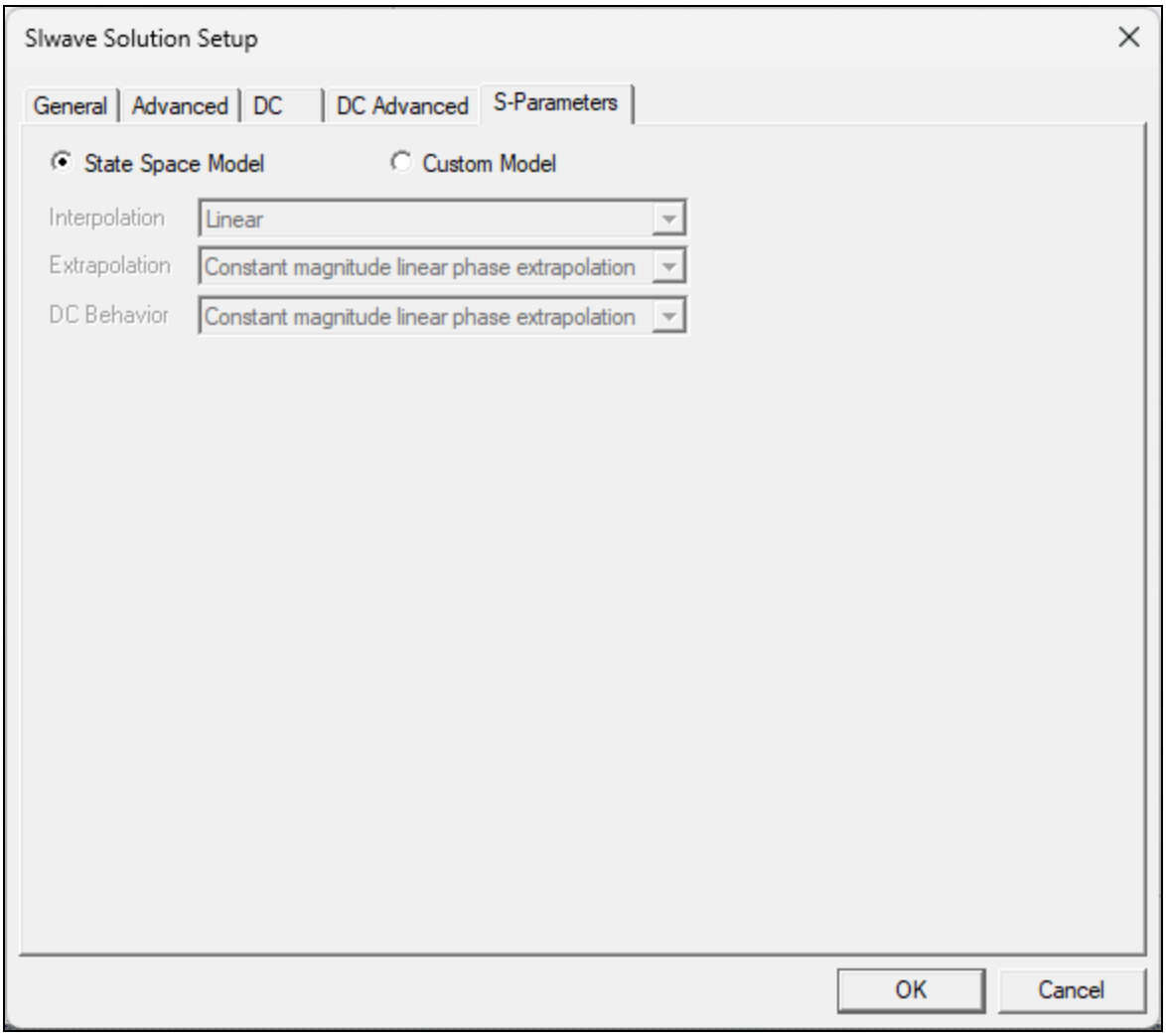
  - **Local Refinement** — this value determines how many triangles are added at each iteration of the adaptive refinement process. The triangles with the highest error is refined. The default value is 20%.
  - **Energy Error** — the default value is 2%.

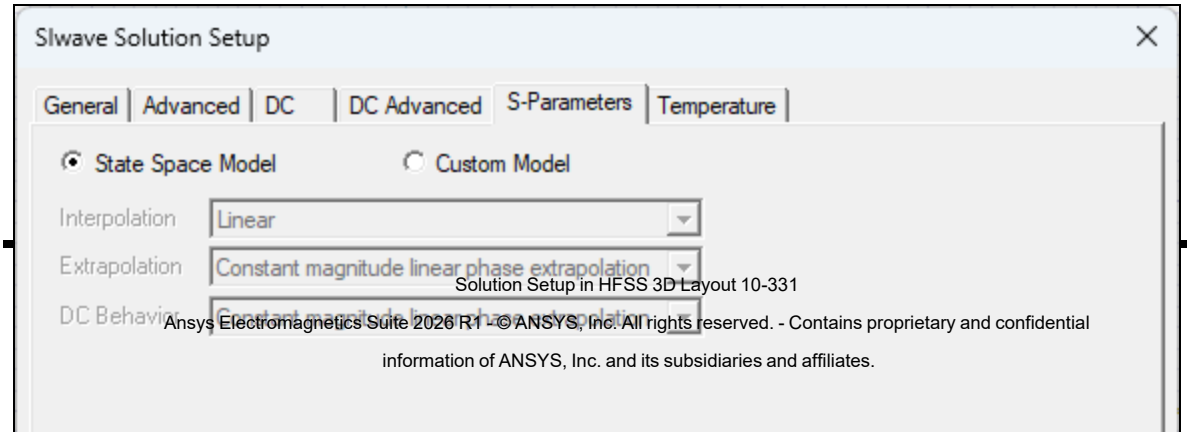
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Refine Mesh Along Bondwires** — check to refine mesh along bondwires.
- **Refine Mesh Along Vias** — check to refine mesh along vias.

S-Parameters Tab

> **Note:**
>
> Parameters on this tab only affect SYZ and Near Field simulations.



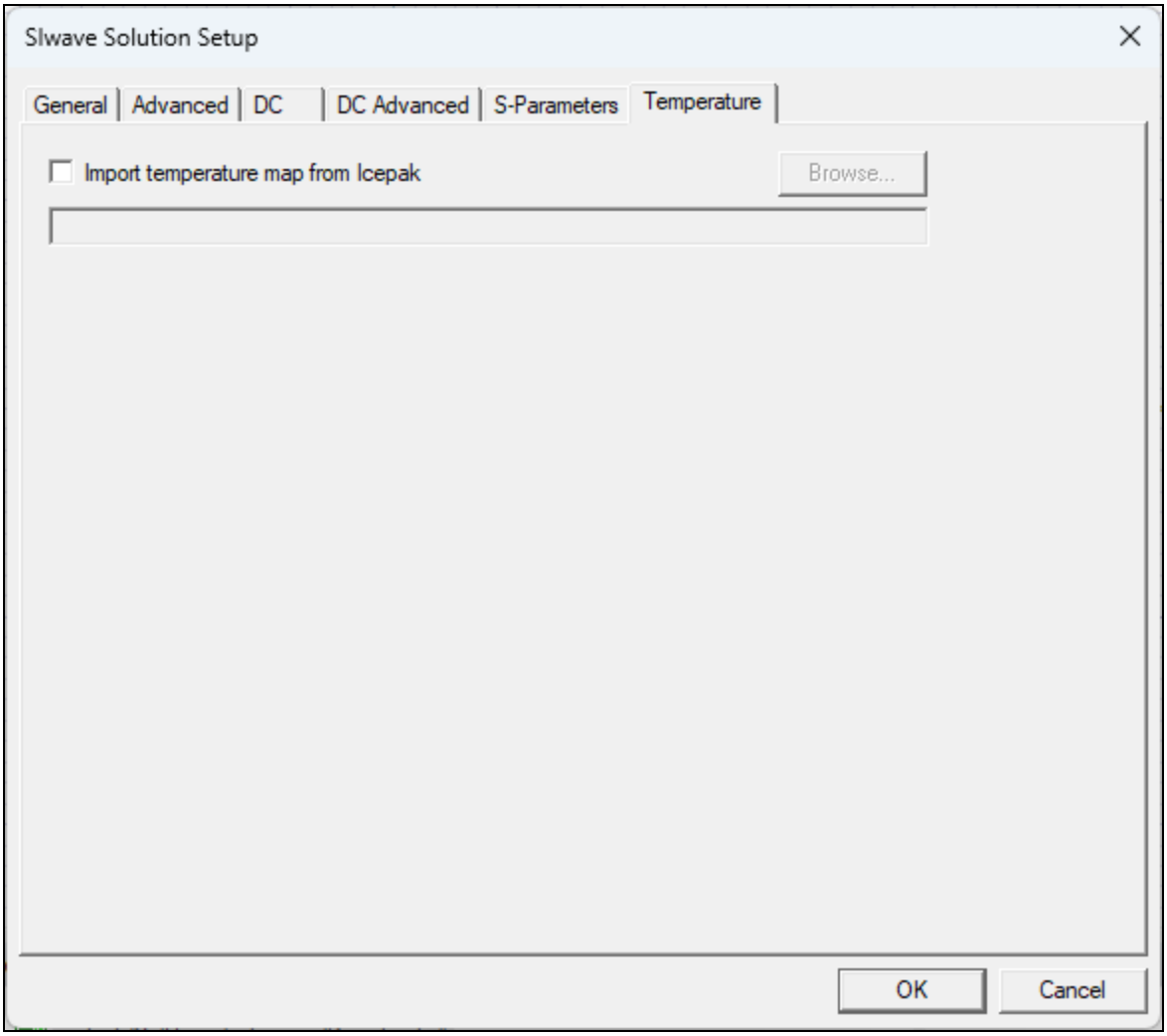


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **S-Parameters** tab, do the following:

- Select **State Space Model** or **Custom Model**. If **Custom Model** is selected, set the following options:
  - **Interpolation** — select **Point**, **Linear**, or **Step** from the drop-down menu.
  - **Extrapolation** — select **Zero padding**, **Same as last point**, **Linear extrapolation from last 2 points**, or **Constant magnitude linear phase extrapolation** from the drop-down menu.
  - **DC Behavior** — select **Zero padding**, **Same as last point**, **Linear extrapolation from last 2 points**, or **Constant magnitude linear phase extrapolation**, **Series Capacitor**, or **Leave all signal lines open circuited** from the drop-down menu.

Temperature Tab



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 **Temperature** tab, check **Import temperature map from Icepak** to select an external Ansys Icepak/Ansys SIwave temperature distribution file, then **Browse** to an appropriate file and select it (e.g., test.sitemp). The temperature data can locally adjust DC conductivity and dielectric permittivity as a function of (x, y, layer) location.

> **Note:**
>
> Materials must have an associated thermal modifier for temperature variation to have an impact.

Running EMI Scanner in HFSS 3D Layout (General Mode Only)

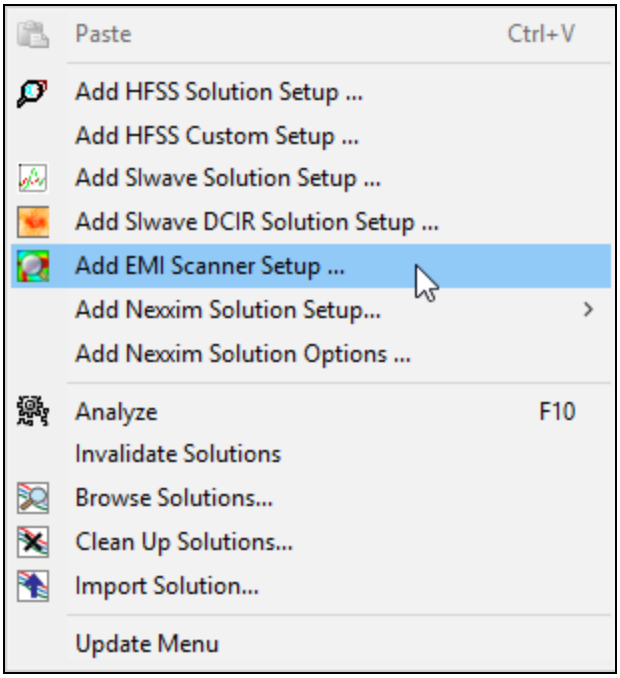
Ansys EMI Scanner identifies design issues that might result in electromagnetic interference problems during operation. Such problems can stem from a compromised power delivery network or from improperly shielded signal nets. This feature, which is based on geometric rule checks, scans complete layouts quickly and can be used to identify subsections of the design requiring more rigorous analysis using the Ansys SIwave™ or Ansys HFSS™ finite element-based solvers.

To set up EMI scanner:

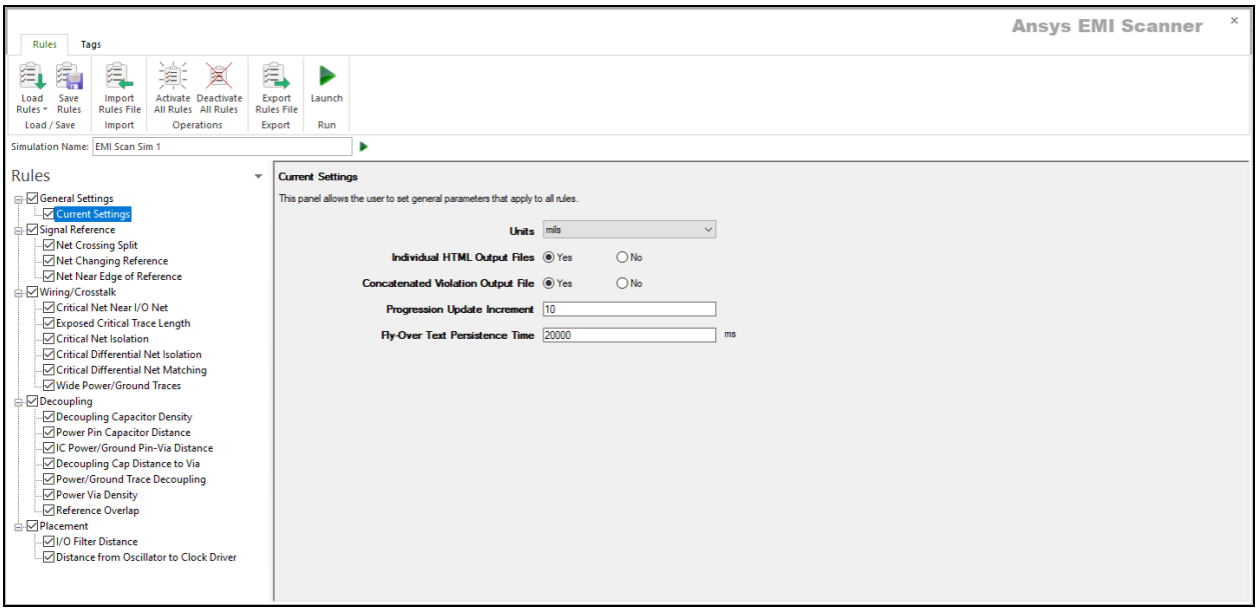
1. From the **Simulation** tab, click **EMI**, or

   From the **Project Manager** window, right-click **Analysis** and select **Add EMI Scanner Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Project Manager** window, double-click **EMI Setup**. to open the **Ansys EMI Scanner** window, on the **Rules** tab.



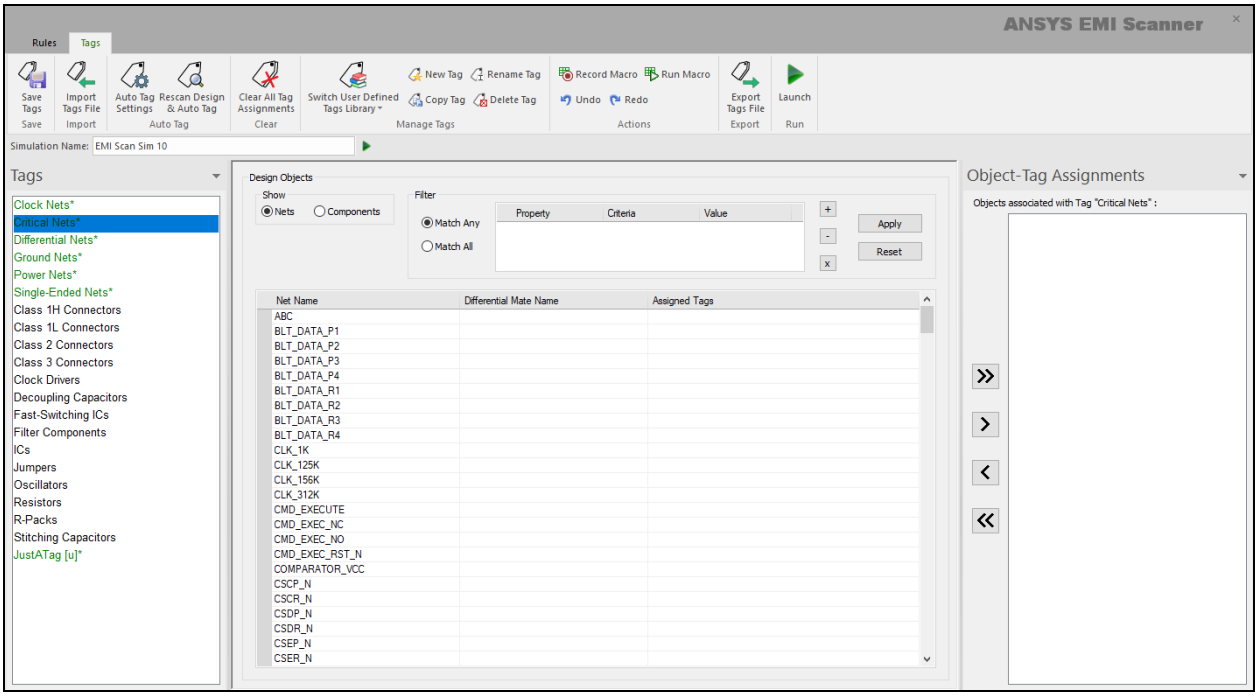
If selected, click the **Simulation Name** field to rename the scan.

3. Set rules, as selected.

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 **Tags** tab.

   The **Tags** tab displays.

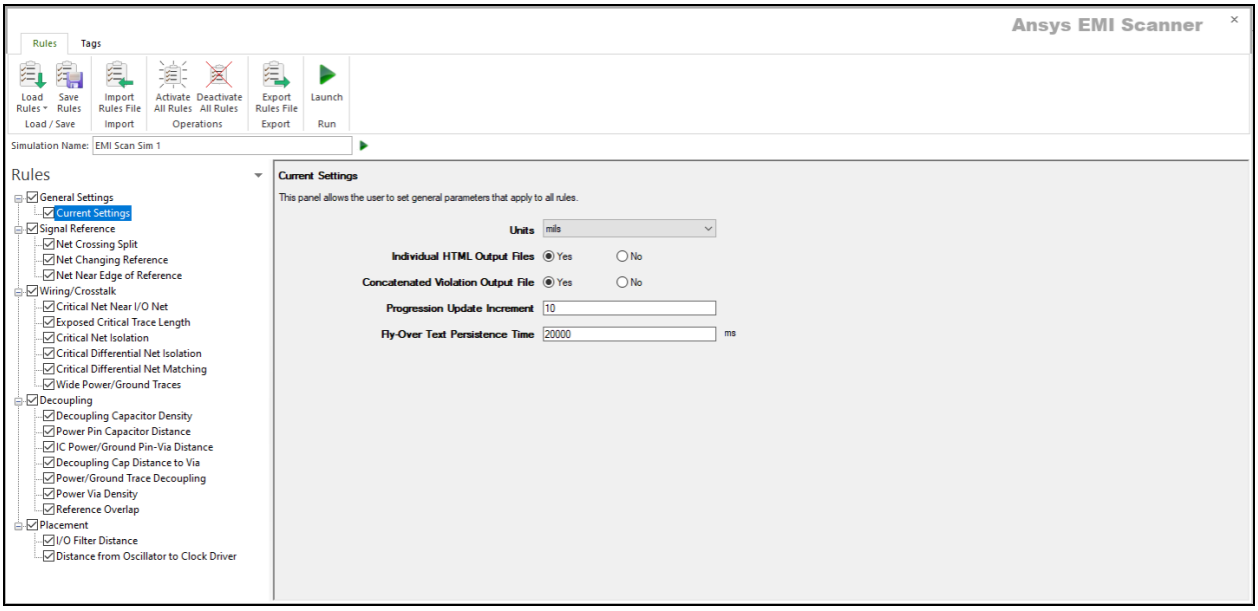


5. Assign tags, as selected.
6. Click **Launch**.

   The **Progress** window updates to show EMI Scanner progress. Simulation Status messages are logged in the **Message Manager**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Rules in EMI Scanner



The EMI Scanner **Rules** tab contains the following buttons:

- **Load Rules** – allows you to select from a series of pre-defined rule sets to load into the Rules pane:
  - **Default EM Rules Profile** – contains the basic rule set for analyzing electromagnetic interference.
  - **Default SI Rules Profile** – contains the basic rule set for analyzing signal integrity.
  - **Default EM+SI Rules Profile** – contains all rules from both the Default EM Rules Profile and the Default SI Rules Profile.
  - **Default Rules Profile (0 - 100MHz)** – contains the recommended rule set for frequencies ranging from 0 - 100MHz.
  - **Default Rules Profile (100MHz - 500MHz)** – contains the recommended rule set for frequencies ranging from 100 - 500MHz.
  - **Default Rules Profile (500MHz - 1000MHz)** – contains the recommended rule set for frequencies ranging from 500 - 1000MHz.
  - **Default Rules Profile (1GHz+)** – contains the recommended rule set for frequencies 1GHz and above.
- **Save Rules** – applies changes to the rule profile.
- **Import Rules File** – allows you to import rules via an XML file.
- **Activate All Rules** – selects all check boxes in the Rules pane.
- **Deactivate All Rules** – deselects all check boxes in the Rules pane.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Export Rules File** – allows you to export the current rule settings in XML format.
- **Launch** – begins the EMI scan.

To set rules for an EMI Scan:

1. Click the **Load Rules** drop-down menu to select a pre-defined rules profile, or click **Import Rules File**.

   The **Rules** pane updates to show all applicable rules for the selected profile.

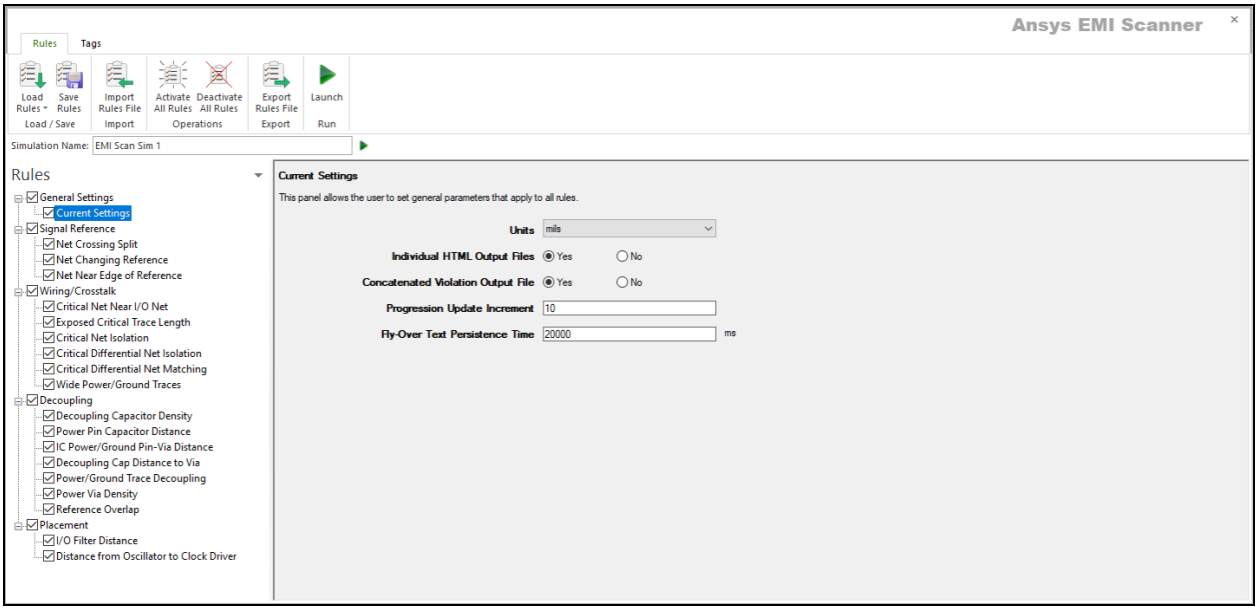
2. From the **Rules** pane, use the check boxes to activate and deactivate rules, as appropriate. Also **Activate All Rules** or **Deactivate All Rules**.
3. Change the General Settings and settings for Signal Reference, Wiring/Crosstalk, Decoupling, Placement, Net Integrity, Via Integrity, and Power Integrity rules, as appropriate.

Applying EMI Scanner General Settings

To change EMI Scanner General Settings:

1. Launch EMI Scanner.

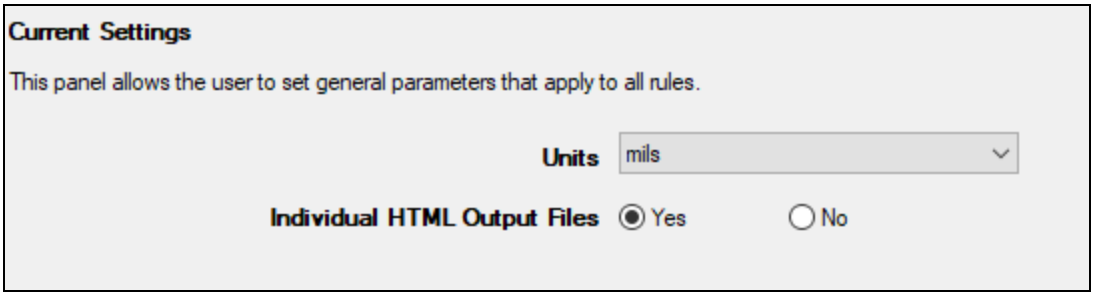
   The **EMI Scanner** window opens, on the **Rules** tab.



2. The **Current Settings** display by default. If they do not, in the Rules pane, select **General Settings** > **Current Settings**.

   The **Current Settings** display.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



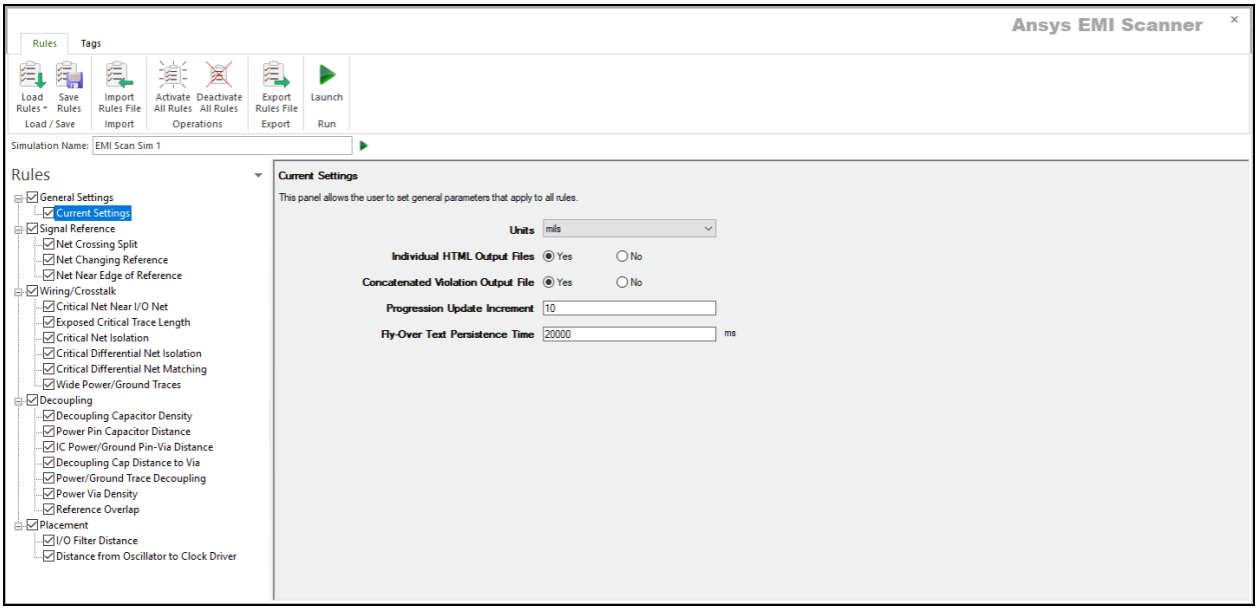
3. From this screen, set the following:
   - **Units** – selects the unit in which other rules are measured.
   - **Individual HTML Output Files** – selects whether to export EMI Scanner results as one large, combined HTML file or as individual HTML files. See Exporting EMI Scanner Results.
4. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

Applying Signal Reference Rules

To change EMI Scanner Signal Reference rules:

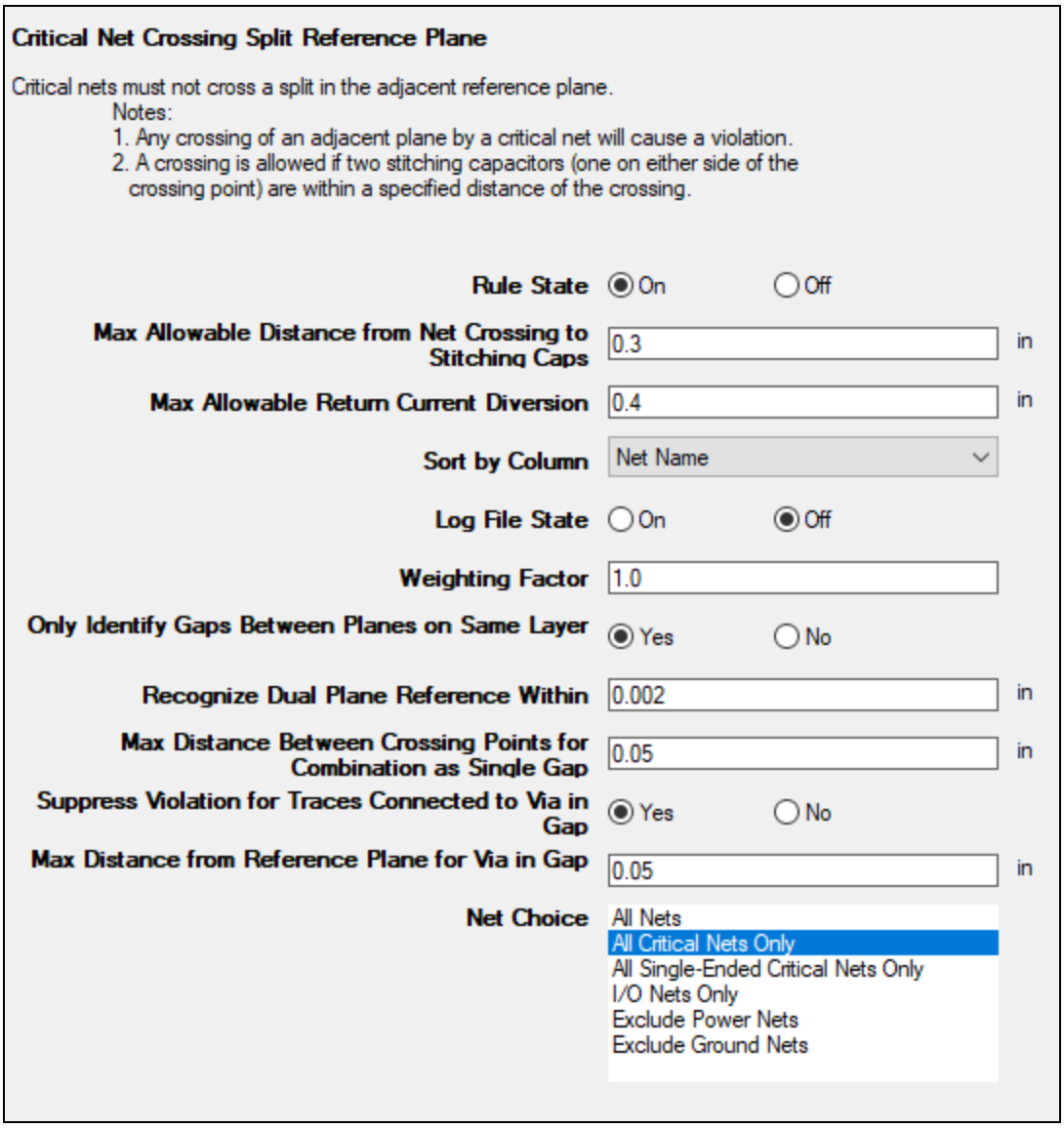
1. Launch EMI Scanner.

   The **EMI Scanner** window opens, on the **Rules** 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. From the **Rules** pane, select one of the three subcategories under **Signal Reference**:
   - **Net Crossing Split** – displays **Critical Net Crossing Split Reference Plane** rule configuration.



This rule states that critical nets must not cross a split in the adjacent reference plane. A cross is allowed only if two stitching capacitors are within a specified distance of the crossing.

From here, set the following:

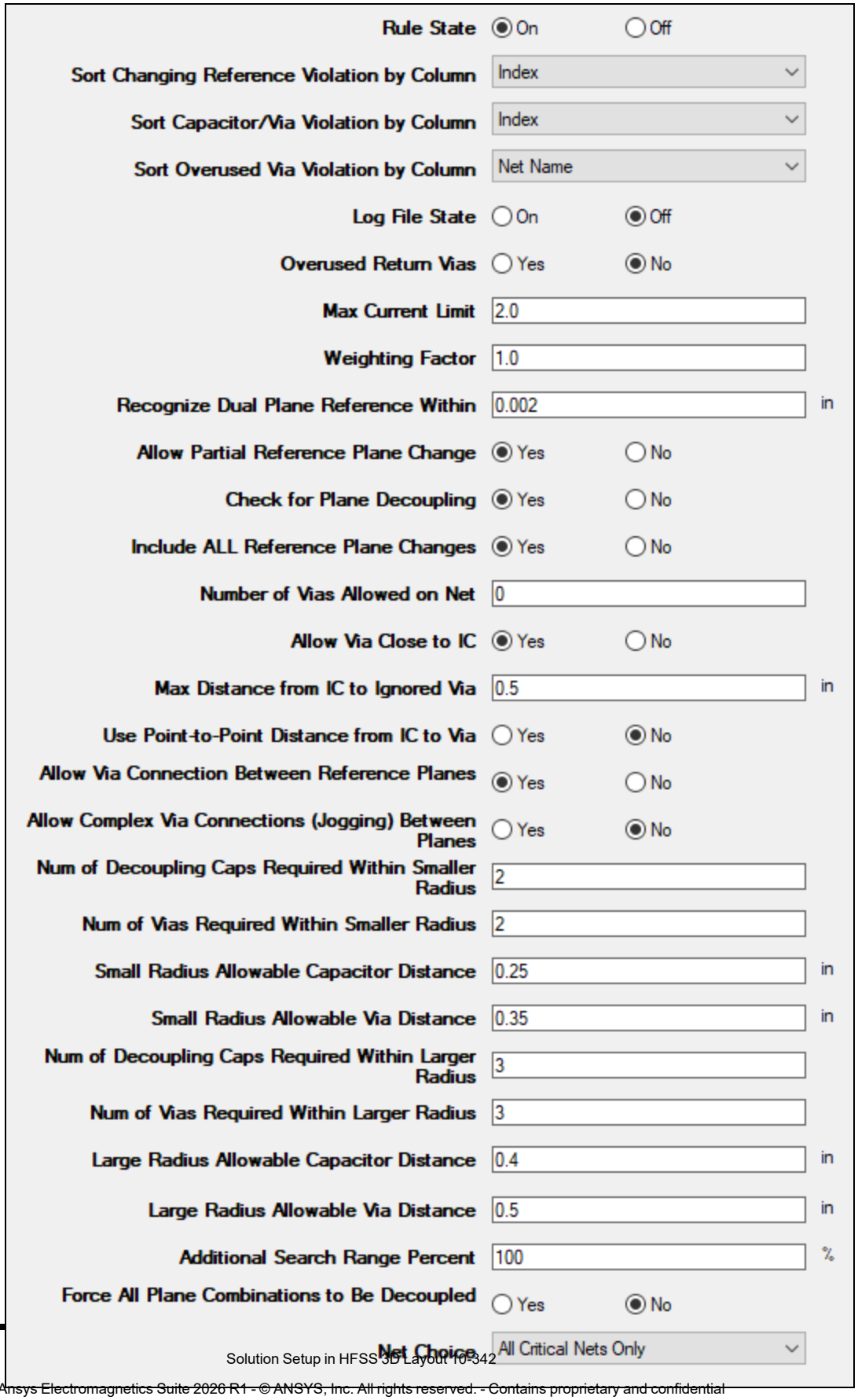
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Rule State** – toggles Net Crossing Split rules on or off.
- **Max Allowable Distance from Net Crossing to Stitching Caps** – sets the maximum distance on the point where the net crosses a split in its reference plane. Two stitching capacitors must be found within this distance.
- **Max Allowable Return Current Diversion** – sets the maximum distance for a return current diversion. If a signal's return current can divert itself around the gap in the reference plane, the total distance of the diversion must be less than this value.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Only Identify Gaps Between Planes on Same Layer** – determines whether gaps are only between planes on the same layer or not. When the rule tries to pair up nearby plane boundary crossing points and identify gaps, this parameter enables or disables a check for layer commonality.
- **Recognize Dual Plane Reference Within** – instructs EMI Scanner to consider both planes as a reference when checking for gap crossing when the distance difference to each plane is less than the specified distance. This allows recognition of both planes as references for strip line traces when the trace is offset on the exact center by a small amount (normally for manufacturing reasons).
- **Max Distance Between Crossing Points for Combination as a Single Gap** – sets the maximum distance for combining crossing points into a single violation (e.g., the entrance and exit of a gap between planes). Too large a value can cause the algorithm to pair up unrelated crossing points, and too small a value can prevent crossing points from being paired up, resulting in more violations. It is recommended to set the value to a reasonable distance based on the board design technology being used (e.g., slightly larger than the typical plane separation within a layer).
- **Suppress Violation for Traces Connected to Via in Gap** – determines whether to ignore trace crossing split/gap when connected to a via within that gap.
- **Max Distance from Reference Plane for Via in Gap** – sets the maximum distance for which via violations within a gap are suppressed. If the trace extends greater than this distance on the edge of the reference plane to the via, its violation are not suppressed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.
- **Net Changing Reference** – displays **Critical Net Changing Reference Plane** rule configuration.

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.

This rule states that critical nets must not change reference planes.

Note:

1. Any change in the reference plane for a critical net causes a violation.
2. Reference plane changing is allowed if two decoupling capacitors (or vias if the planes share the same net name) are within a specified radial distance on the signal via where the signal changes reference planes.
3. Reference plane changing is allowed if three decoupling capacitors (or vias if the planes share the same net name) are within a specified radial distance on the signal via where the signal changes reference planes.
4. Reference plane changing can be ignored when the initial via is within a specified distance from its IC pin (transmit or receive).

From here, set the following:

- **Rule State** – toggles Net Changing Reference rules on or off.
- **Sort Changing Reference Violation by Column** – specifies the column by which Changing Reference Violation results are sorted.
- **Sort Capacitor/Via Violation by Column** – specifies the column by which Capacitor/Via Violation results are sorted.
- **Sort Overused Via Violation by Column** – specifies the column by which Overused Via Violation results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Overused Return Vias** – determines whether EMI Scanner checks for overused return vias.
- **Max Current Limit** – specifies the maximum number of signal currents that a via should carry. Use in conjunction with **Overused Return Vias**.
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Recognize Dual Plane Reference Within** – instructs EMI Scanner to consider both planes as a reference when checking for gap crossing when the distance difference to each plane is less than the specified distance. This allows recognition of both planes as references for strip line traces when the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

trace is offset on the exact center by a small amount (normally for manufacturing reasons).

- **Allow Partial Reference Plane Change** – instructs EMI Scanner to ignore a violation if one of the two reference planes remains a reference plane on both sides of the via transition.
- **Check for Plane Decoupling** – instructs EMI Scanner to ignore a violation if a decoupling capacitor exists between the two reference planes (entering reference plane and leaving reference plane) and is within a specified distance of the signal via.
- **Include ALL Reference Plane Changes** – instructs EMI Scanner whether to display all reference plane changes. When there are two associated reference planes for the entry to the via, and two different associated reference planes for the trace leaving the via, selecting 'yes' shows two violations for this transition. If 'no' is selected and a decoupling capacitor exists between one of the entering reference planes and one of the leaving reference planes (in the specified distance), no violation are reported, even though the two other reference planes do not have any connection.
- **Number of Vias Allowed on Net** – specifies the number of via reference plane changes that are allowed on critical nets. If no vias are selected except the 'escape' via close to the IC, set this parameter to zero, and set **Allow Via Close to IC** to 'yes'.
- **Allow Via Close to IC** – determines whether to ignore vias that are close to the IC and are required to get a trace on the surface to an inner layer.
- **Max Distance from IC to Ignored Via** – specifies the maximum distance between IC and via at which vias is ignored.
- **Use Point-to-Point Distance from IC to Via** – if 'yes' is selected, the actual trace path length is NOT calculated. This option allows EMI Scanner to check rules faster, but violations may be ignored incorrectly.
- **Allow Via Connection Between Reference Planes** – allows a via to connect reference planes (to provide return current path) when the two reference planes share the same net name.
- **Allow Complex Via Connections (Jogging) Between Planes** – allows a set of vias (e.g., 'blind' vias that do not extend through all layers) to connect with reference planes (to provide return current path) when the two reference planes share a net name. This advanced checking requires extra processing time because all vias found in the larger via radius are considered as potential paths.
- **Num of Decoupling Caps Required Within Smaller Radius** – when **Allow Partial Reference Plane Change** is selected, a return current path (when a critical net changes reference planes) may be provided with either capacitors

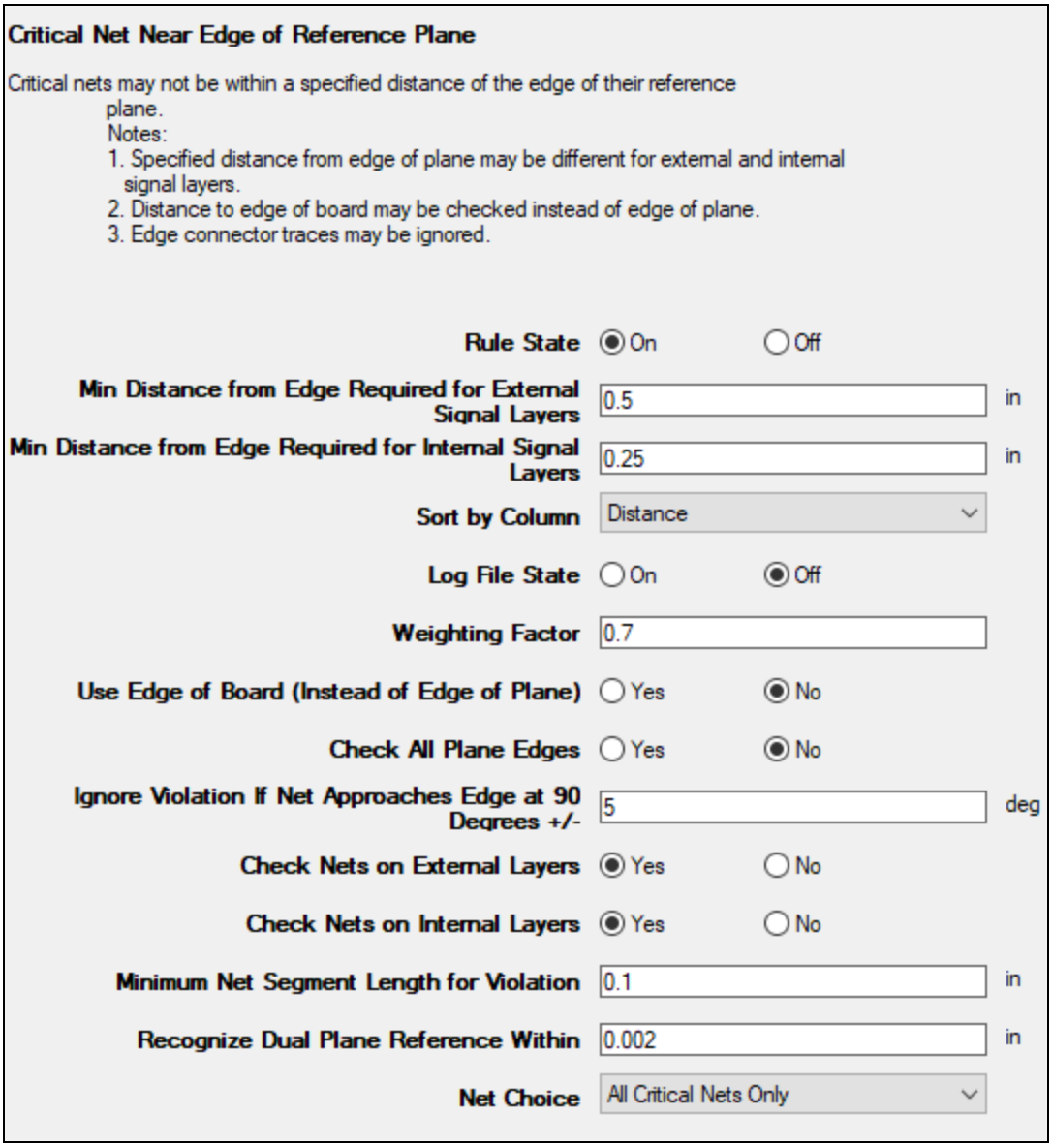
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or vias. This setting specifies the number of capacitors required in the smaller radius.

- **Num of Vias Required Within Smaller Radius** – when **Allow Partial Reference Plane Change** is selected, a return current path (when a critical net changes reference planes) may be provided with either capacitors or vias. This setting specifies the number of vias required in the smaller radius. A via between planes is a lower impedance/inductance connection than a decoupling capacitor, so you may want to specify a different setting.
- **Small Radius Allowable Capacitor Distance** – specifies the smaller radius used with the **Num of Decoupling Caps Required Within Smaller Radius** setting.
- **Small Radius Allowable Via Distance** – specifies the smaller radius used with the **Num of Vias Required Within Smaller Radius** setting.
- **Num of Decoupling Caps Required Within Larger Radius** – if a smaller radius is not specified, this allows specification of a larger radius for a larger number of decoupling capacitors.
- **Num of Vias Required Within Larger Radius** – if a smaller radius is not specified, this allows specification of a larger radius for a larger number of vias. A via between planes is a lower impedance/inductance connection then a decoupling capacitor, so you may want to specify a different setting.
- **Large Radius Allowable Capacitor Distance** – specifies the larger radius used with the **Num of Decoupling Caps Required Within Larger Radius** setting.
- **Large Radius Allowable Via Distance** – specifies the larger radius used with the **Num of Vias Required Within Larger Radius** setting.
- **Additional Search Range Percent** – specifies an additional search range beyond the larger radius for additional capacitors (e.g., if the larger radius is set to 500mils and the Additional Search Range Percent is 100%, EMI Scanner reports capacitors within 1000mils when a violation occurs).
- **Force All Plane Combinations to Be Decoupled** – determines whether to decouple all reference plane combinations. See Technical Note.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Net Near Edge of Reference** – displays **Critical Net Near Edge of Reference Plane** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This rule states that critical nets must not be within a specified distance of the edge of their reference plane.

From here, set the following:

- **Rule State** – toggles Net Near Edge of Reference rules on or off.
- **Min Distance from Edge Required for External Signal Layers** – specifies the minimum distance that must exist between the edge of the reference plane and a critical signal net when the net is on an external layer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Min Distance from Edge Required for Internal Signal Layers** – specifies the minimum distance that must exist between the edge of the reference plane and a critical signal net when the net is on an internal layer.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Use Edge of Board (Instead of Edge of Plane)** – allows you to check distance from critical nets to the edge of the board instead of the edge of the reference plane. This can be useful if the board is only partially complete and all reference planes are not yet completed.
- **Check All Plane Edges** – checks the distance from nets to any edge of their reference plane, rather than limiting the check to reference plane edges near the edge of the board
- **Ignore Violation If Net Approaches Edge at 90 Degrees +/-** – EMI scanner ignores edge connectors that approach the edge of the reference plane or board at a 90 degree angle, tolerating approaches slightly off a perfect 90 degrees.
- **Check Nets on External Layers** – specifies whether nets on external layers should be checked.
- **Check Nets on Internal Layers** – specifies whether nets on internal layers should be checked.
- **Minimum Net Segment Length for Violation** – specifies the minimum net segment length before EMI Scanner reports a violation if the net is in the specified distance on the edge of the reference plane or board.
- **Recognize Dual Plane Reference Within** – instructs EMI Scanner to consider both planes as a reference when checking for traces too close to the edge of the reference planes when the distance difference to each plane is less than the specified distance. This allows recognition of both planes as references for strip line traces when the trace is offset on the exact center by a small amount (normally for manufacturing reasons).
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

3. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

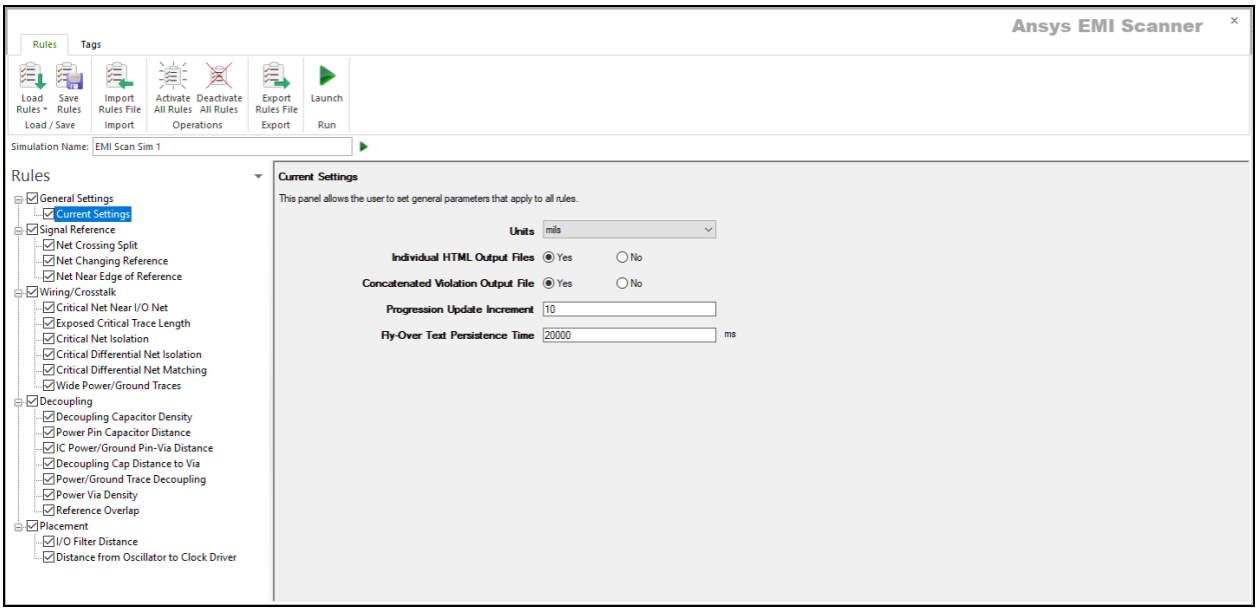
- You can display a description of a rule parameter at any time as with the cursor over it.
- Units of measure are set in General Settings.

Applying Wiring/Crosstalk Rules

To change EMI Scanner Wiring/Crosstalk rules:

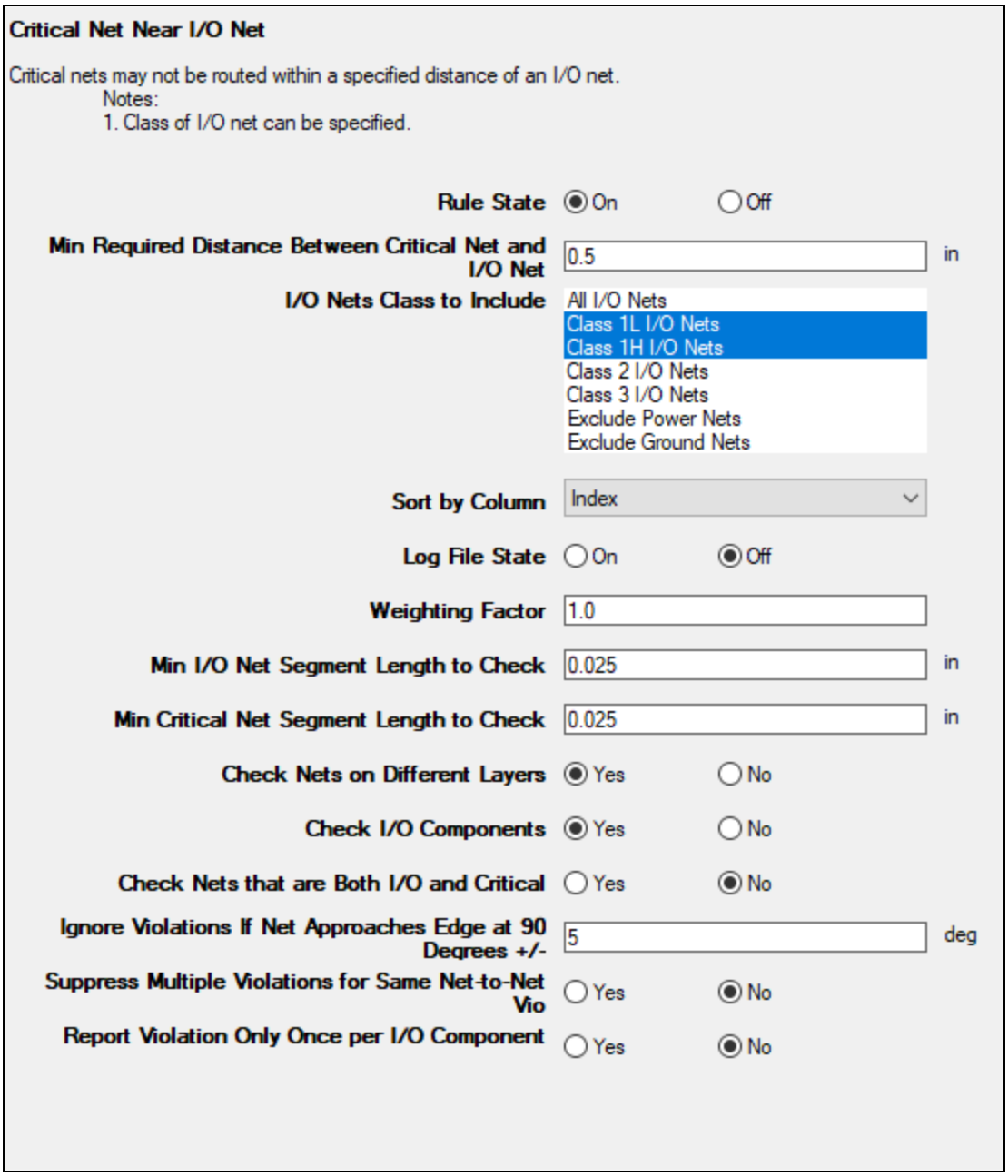
1. Launch EMI Scanner.

   The **EMI Scanner** window opens, on the **Rules** 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. From the **Rules** pane, select one of the six subcategories under **Wiring/Crosstalk**:
   - **Critical Net Near I/O Net** – displays **Critical Net Near I/O Net** rule configuration.



This rule states that critical nets may not be routed within a specified distance from an I/O net.

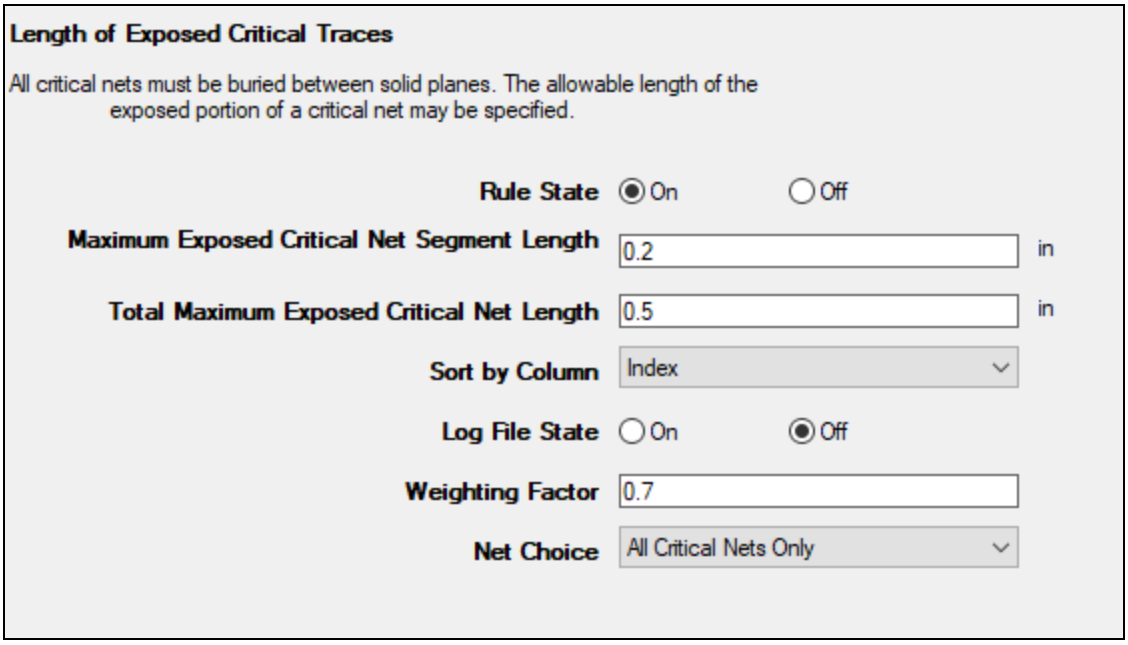
From here, set the following:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Rule State** – toggles Critical Net Near I/O Net rules on or off.
- **Min Required Distance Between Critical Net and I/O Net** – specifies the minimum distance between all critical nets and all I/O nets.
- **I/O Nets Class to Include** – specifies the type(s) of I/O Nets to include. Select one, or **Ctrl**+click to select multiple types. See Technical Note.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Min I/O Net Segment Length to Check** – specifies the smallest I/O net segment length to check for violations. EMI Scanner ignores All I/O net segments smaller than this length.
- **Min Critical Net Segment Length to Check** – specifies the smallest critical net segment length to check for violations. EMI Scanner ignores all critical net segments smaller than this length.
- **Check Nets on Different Layers** – determines whether to check the distance between I/O nets and critical nets when these nets are on different layers and between the same pair of planes.
- **Check I/O Components** – determines whether any I/O component that is in the specified distance from a critical net causes a violation.
- **Check Nets that are Both I/O and Critical** – determines whether nets labeled as both critical and I/O should be checked for distance from other I/O nets.
- **Ignore Violations If Net Approaches Edge at 90 Degrees +/-** – EMI scanner ignores edge connectors that approach the edge of the reference plane or board at a 90 degree angle, tolerating approaches slightly off a perfect 90 degrees.
- **Suppress Multiple Violations for Same Net-to-Net Ratio** – determines whether to suppress multiple violation reporting when multiple segments on a critical net infringe on the same I/O net.
- **Report Violation Only Once per I/O Component** – reduces run time by checking only for violations between critical nets and I/O components when no other violation has been found for an I/O component.

- **Exposed Critical Trace Length** – displays **Length of Exposed Critical Traces** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



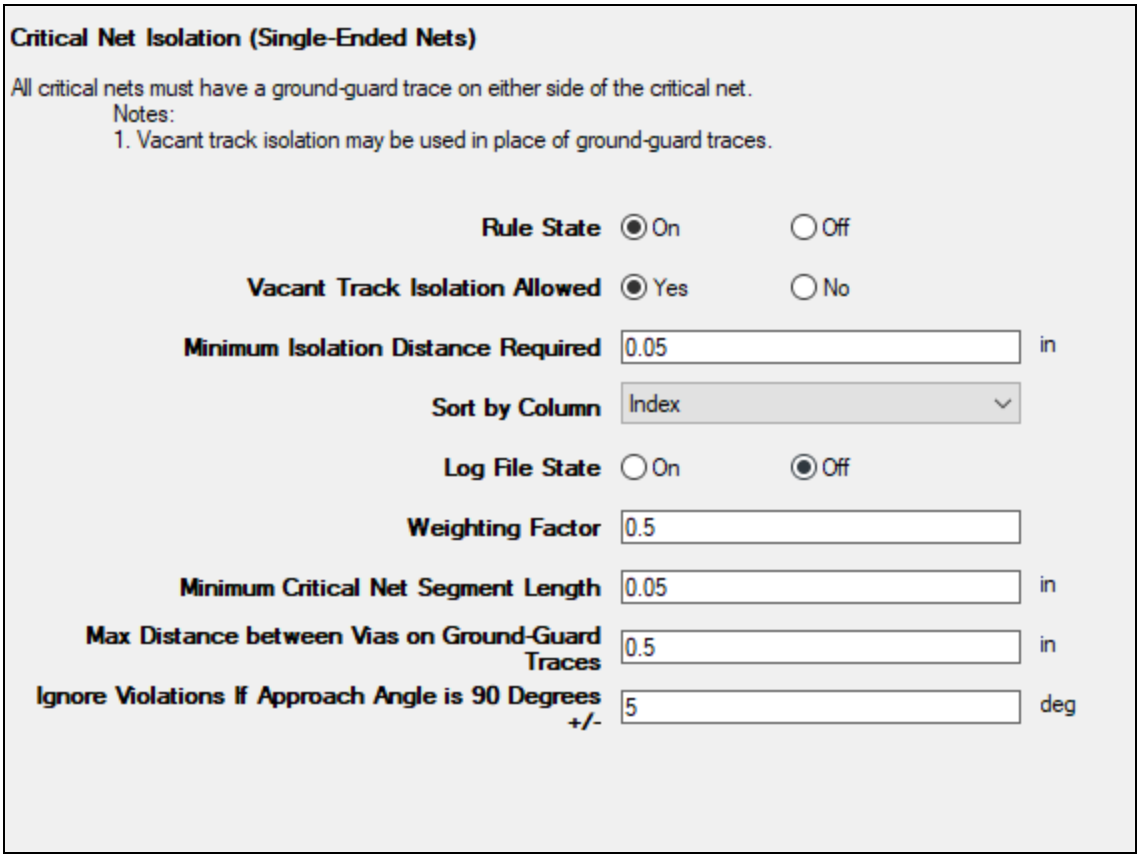
This rule states that all critical nets must be buried between solid planes.

From here, set the following:

- **Rule State** – toggles Exposed Critical Trace Length rules on or off.
- **Maximum Exposed Critical Net Segment Length** – specifies the maximum critical net segment length allowed on external (exposed) layers.
- **Total Maximum Exposed Critical Net Length** – specifies the total maximum critical net length allowed on external (exposed) layers (e.g., multiple segments on a critical net may be exposed, but short enough not to cause a violation individually).
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Critical Net Isolation** – displays **Critical Net Isolation (Single-Ended Nets)** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This rule states that all critical nets must have empty tracks or a "ground-guard" trace on either side of the critical net.

From here, set the following:

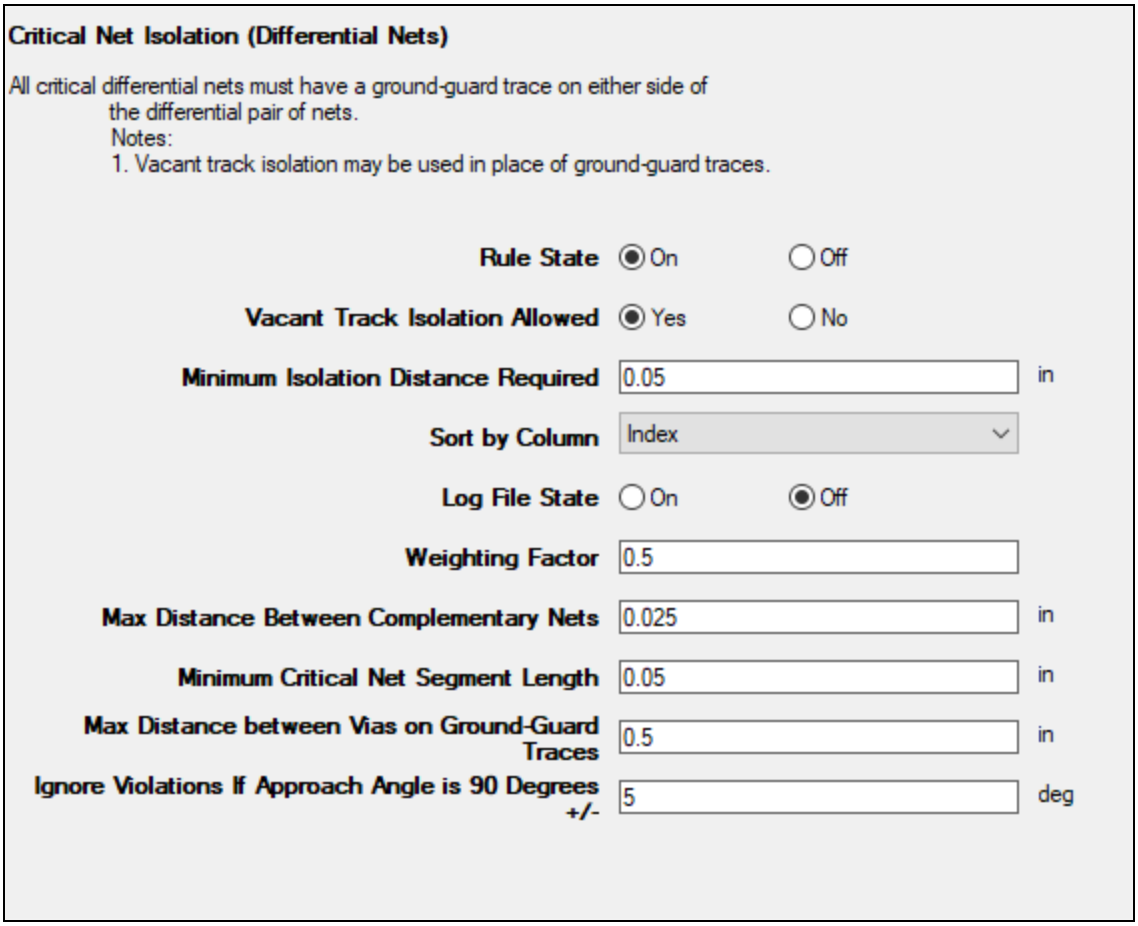
- **Rule State** – toggles Critical Net Isolation rules on or off.
- **Vacant Track Isolation Allowed** – specifies whether vacant tracks (empty space) may be used in place of ground-guard traces.
- **Minimum Isolation Distance Required** – selects the minimum edge-to-edge distance required between critical nets and all other nets before EMI Scanner reports a violation. Used in coordination with **Vacant Track Isolation Allowed**.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

- **Minimum Critical Net Segment Length** – specifies the minimum critical net segment length before EMI Scanner reports a violation. Segments shorter than this length is ignored.
- **Max Distance Between Vias on Ground-Guard Traces** – specifies the maximum distance between vias on ground-guard traces.
- **Ignore Violations If Approach Angle is 90 Degrees +/-** – EMI scanner ignores edge connectors that approach the edge of the reference plane or board at a 90 degree angle, tolerating approaches slightly off a perfect 90 degrees.

- **Critical Differential Net Isolation** – displays **Critical Net Isolation (Differential Nets)** rule configuration.



This rule states that all differential critical nets must have empty tracks or a "ground-guard" trace on either side of the differential pair.

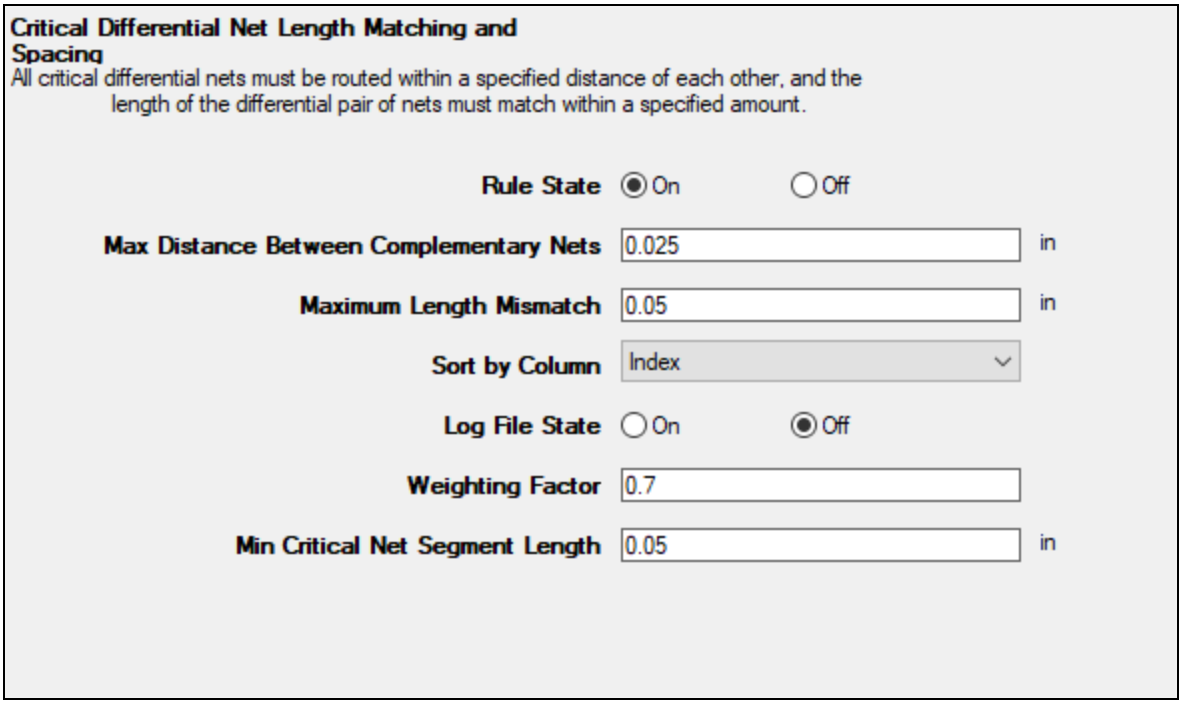
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From here, set the following:

- **Rule State** – toggles Critical Differential Net Isolation rules on or off.
- **Vacant Track Isolation Allowed** – determines whether vacant tracks (empty space) may be used in place of ground-guard traces.
- **Minimum Isolation Distance Required** – when **Vacant Track Isolation Allowed** is selected, this setting determines the minimum edge-to-edge distance required between critical nets and all other nets before EMI Scanner reports a violation.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Max Distance Between Complementary Nets** – sets the maximum spacing between a pair of critical nets (creating a critical differential pair).
- **Minimum Critical Net Segment Length** – allows you to ignore short critical net segments by specifying the minimum critical net segment length before EMI Scanner reports a violation.
- **Max Distance between Vias on Ground-Guard Traces** – specifies the maximum distance between vias on ground-guard traces.
- **Ignore Violations If Approach Angle is 90 Degrees +/-** – EMI scanner ignores edge connectors that approach the edge of the reference plane or board at a 90 degree angle, tolerating approaches slightly off a perfect 90 degrees.

- **Critical Differential Net Matching** – displays **Critical Differential Net Length Matching and Spacing** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



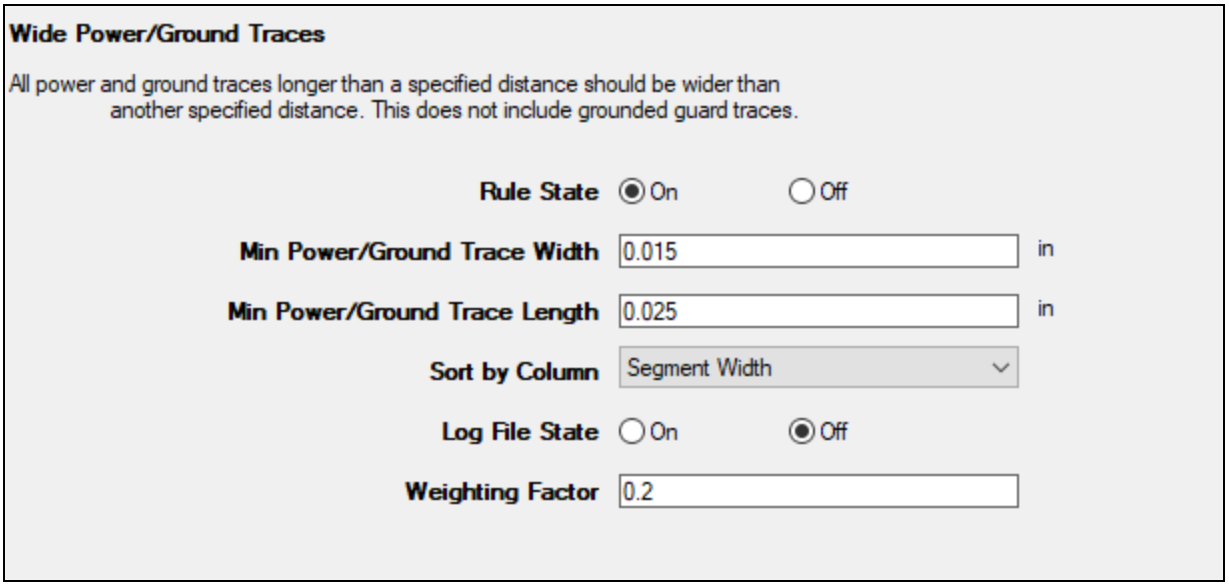
This rule states that all critical nets must be routed within a specified distance of each other, and the length of the differential pair must match within a specified amount.

From here, set the following:

- **Rule State** – toggles Critical Differential Net Matching rules on or off.
- **Max Distance Between Complementary Nets** – sets the maximum spacing between a pair of critical nets (creating a critical differential pair).
- **Maximum Length Mismatch** – specifies the maximum difference in length between a critical differential pair of nets.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Min Critical Net Segment Length** – allows you to ignore short critical net segments by specifying the minimum critical net segment length before EMI Scanner reports a violation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Wide Power/Ground Traces** – displays **Wide Power/Ground Traces** rule configuration.



This rule states that all power and ground traces longer than a specified distance must be wider than a specified amount.

From here, set the following:

- **Rule State** – toggles Wide Power/Ground Traces rules on or off.
- **Min Power/Ground Trace Width** – sets a minimum width limit for all power/ground traces (except grounded guard traces).
- **Min Power/Ground Trace Length** – specifies the smallest trace length to check for violations. EMI Scanner ignores traces smaller than this length.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

3. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

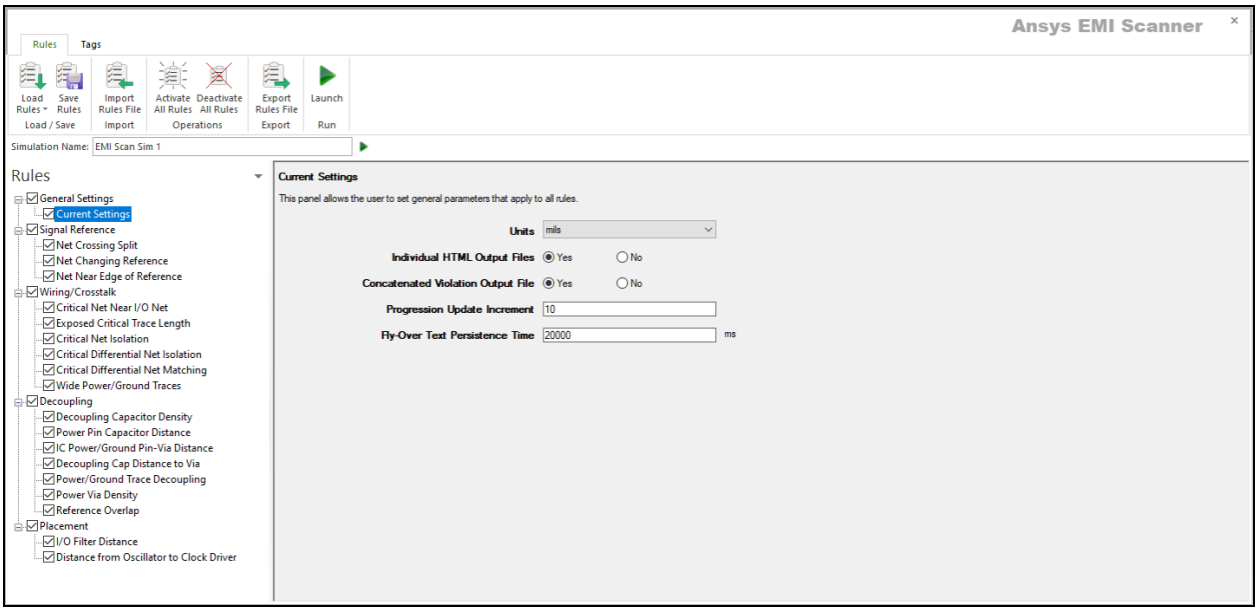
- You can display a description of a rule parameter at any time as with the cursor over it.
- Units of measure are set in General Settings.

Applying Decoupling Rules

To change EMI Scanner Decoupling rules:

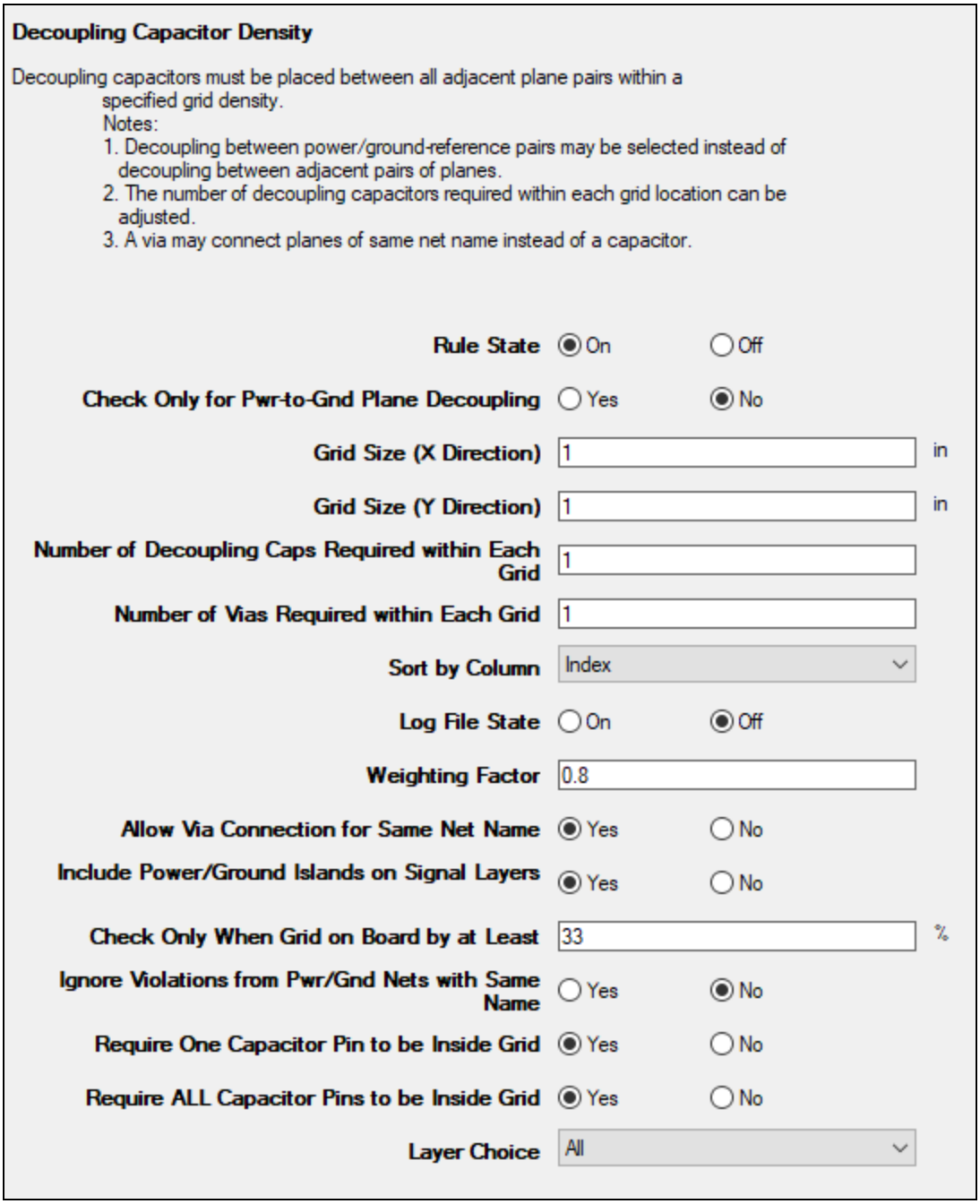
1. Launch EMI Scanner.

   The **EMI Scanner** window opens, on the **Rules** tab.



2. From the Rules pane, select one of the six subcategories under **Decoupling**:
   - **Decoupling Capacitor Density** – displays **Decoupling Capacitor Density** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This rule states that decoupling capacitors must be placed between all adjacent plane pairs within a specified grid density.

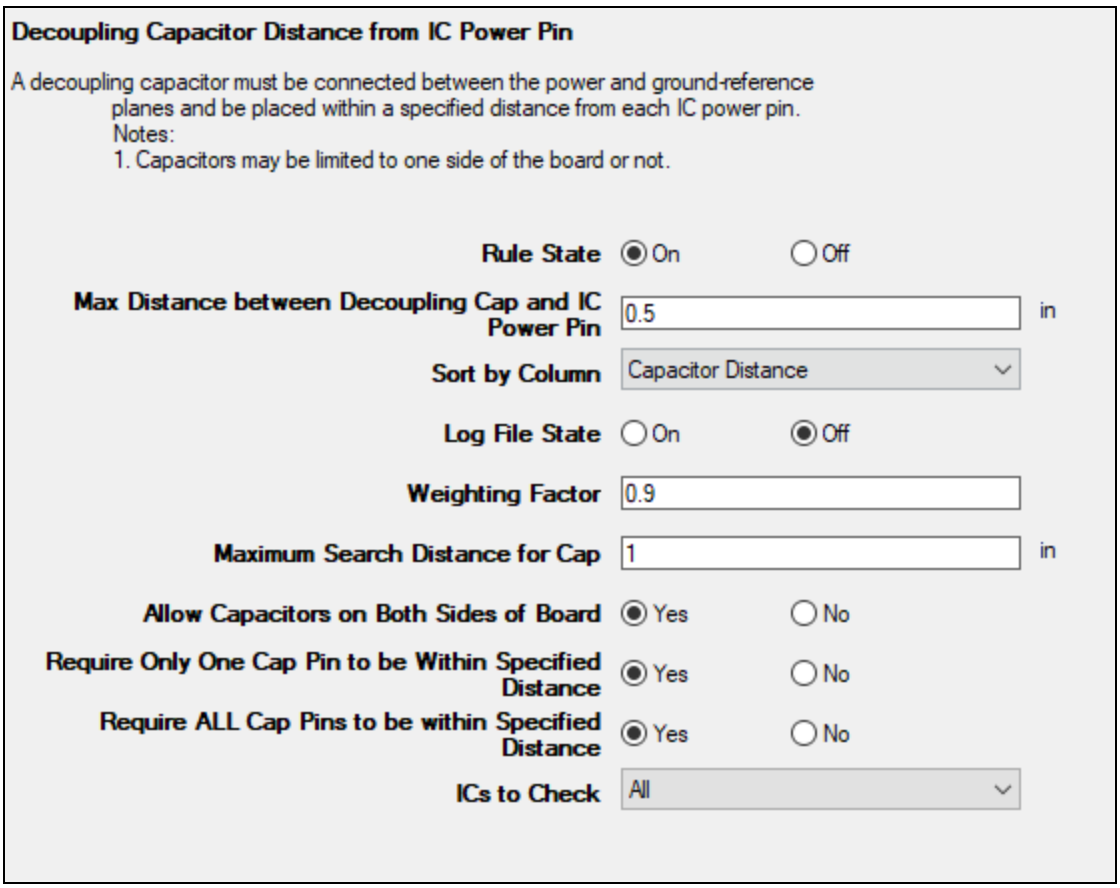
From here, set the following:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Rule State** – toggles Decoupling Capacitor Density rules on or off.
- **Check Only for Pwr-to-Gnd Plane Decoupling** – sets whether to ignore plane-to-plane decoupling and check only for decoupling capacitors between power and ground planes.
- **Grid Size (X Direction)** – specifies the size of the grid used to check for decoupling capacitors. Used in conjunction with **Grid Size (Y Direction)**.
- **Grid Size (Y Direction)** – specifies the size of the grid used to check for decoupling capacitors. Used in conjunction with **Grid Size (X Direction)**.
- **Number of Decoupling Caps Required within Each Grid** – specifies the number of decoupling capacitors connecting between planes that are required within each grid location.
- **Number of Vias Required within Each Grid** – specifies the number of vias connecting between planes that are required within each grid location. You must select **Allow Via Connection for Same Net Name** to use this option.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Allow Via Connection for Same Net Name** – determines whether EMI Scanner allows a via in place of a decoupling capacitor when the two planes are the same net name.
- **Include Power/Ground Islands on Signal Layers** – specifies whether to check for decoupling on power/ground islands.
- **Check Only When Grid on Board by at Least** – specifies the percentage of a grid that must be on the board in order for it to be checked.
- **Ignore Violations from Pwr/Gnd Nets with Same Name** – determines that only one plane of a given name is checked for violations (e.g., if two planes are "gnd" and a violation occurs between gnd and another layer, only one violation is reported).
- **Require One Capacitor Pin to be Inside Grid** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **Require ALL Capacitor Pins to be Inside Grid** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **Layer Choice** – specifies the layers to include in this rule analysis.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Power Pin Capacitor Distance** – displays **Decoupling Capacitor Distance from IC Power Pin** rule configuration.



This rule states that a decoupling capacitor must be connected between the power and ground-reference planes and be placed within a specified distance of each IC power pin.

From here, set the following:

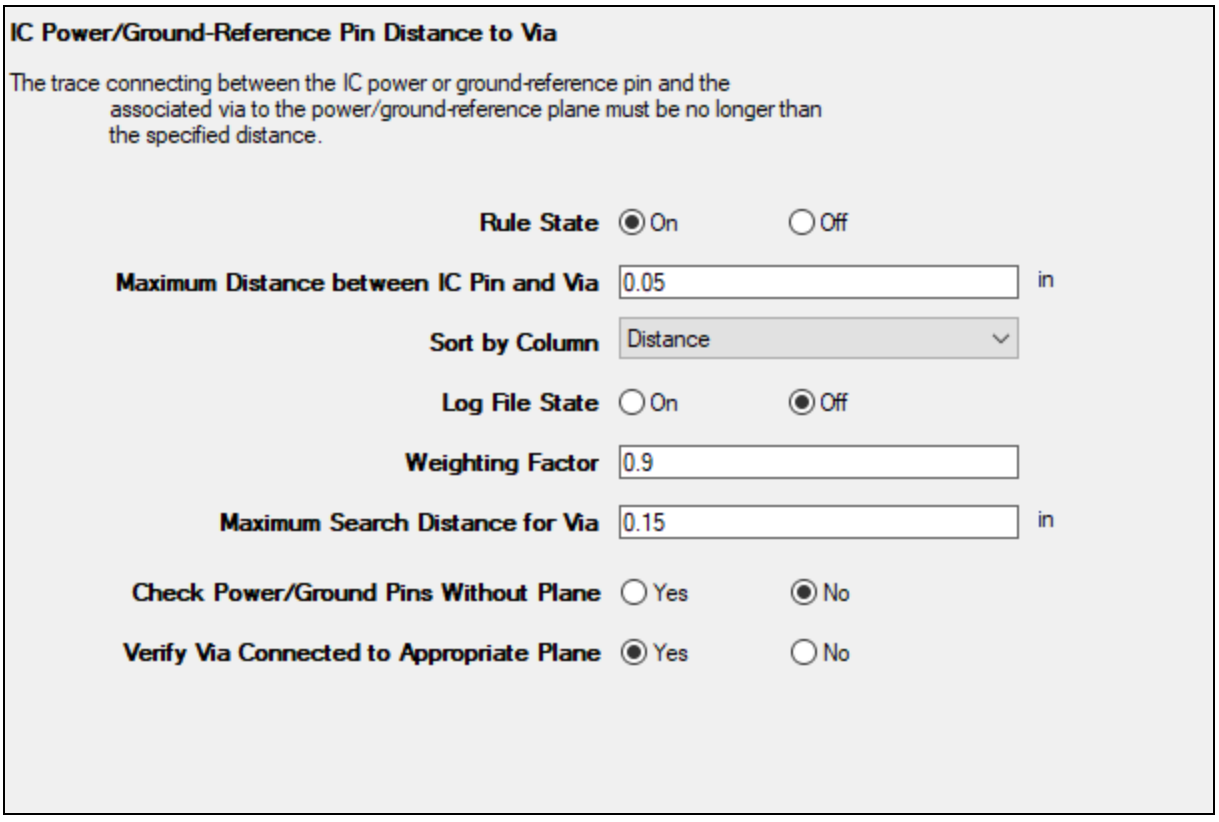
- **Rule State** – toggles Power Pin Capacitor Distance rules on or off.
- **Max Distance between Decoupling Cap and IC Power Pin** – specifies the maximum distance between an IC power pin and its decoupling capacitor.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

- **Maximum Search Distance for Cap** – specifies a maximum search distance for a capacitor. A larger search box increases the chances of EMI Scanner finding capacitors, but can slow processing time.
- **Allow Capacitors on Both Sides of Board** – specifies whether to include capacitors on both sides of the board.
- **Require Only One Cap Pin to be Within Specified Distance** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **Require ALL Cap Pins to be within Specified Distance** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **ICs to Check** – selects the types of components to include in this rule analysis.

- **IC Power/Ground Pin-Via Distance** – displays **IC Power/Ground-Reference Pin Distance to Via** rule configuration.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

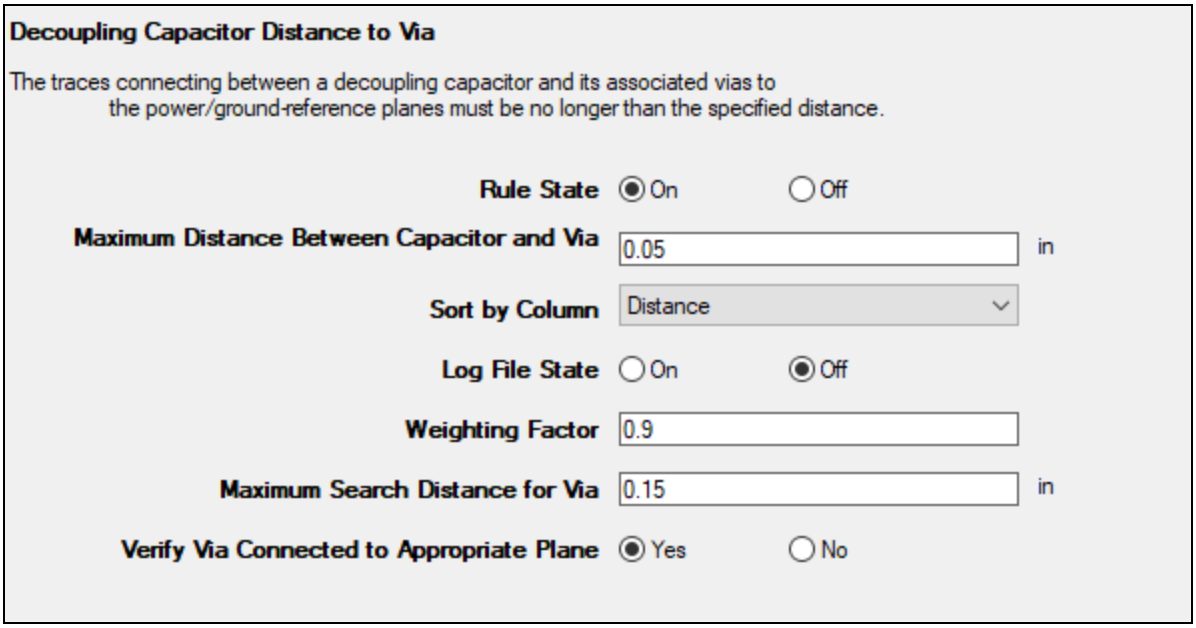
This rule states the trace connecting the IC power/ground reference pin to its associated via to the power/ground-reference plane must be no longer than a specified distance.

From here, set the following:

- **Rule State** – toggles IC Power/Ground Pin-Via Distance rules on or off.
- **Maximum Distance between IC Pin and Via** – specifies the maximum distance between the center of the IC power or ground-reference pin and the center of the via to the power or ground-reference plane.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Maximum Search Distance for Via** – specifies a maximum search distance for a via when a violation is found. This allows you to spot vias that are barely outside the original requirement and therefore not serious violations.
- **Check Power/Ground Pins Without Plane** – checks the distance between the IC power/ground pin and its via when the power/ground plane has not been completed yet.
- **Verify Via Connected to Appropriate Plane** – selecting 'no' speeds up EMI Scanner by not verifying the via connects to the selected plane (shape).

- **Decoupling Cap Distance to Via** – displays **Decoupling Capacitor Distance to Via** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



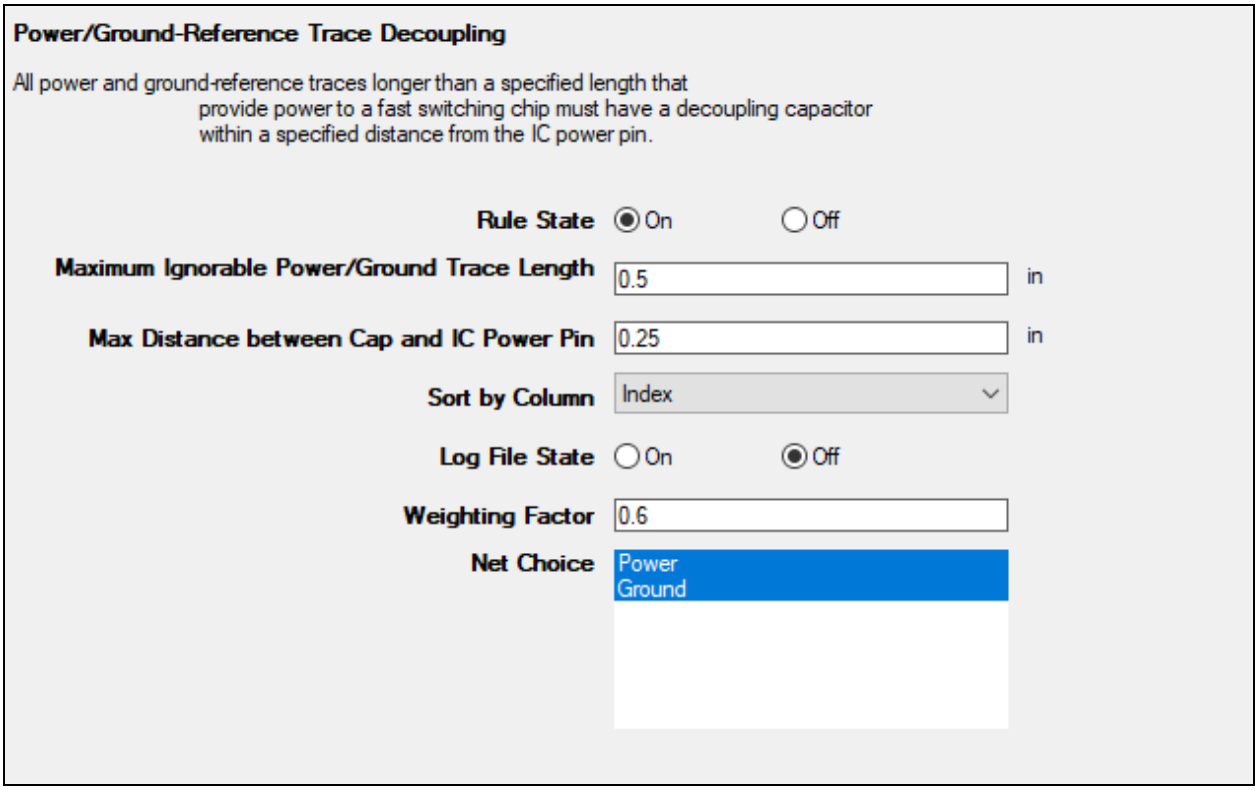
This rule states the trace connecting a decoupling capacitor to its associated via to the power/ground-reference plane must be no longer than the specified distance.

From here, set the following:

- **Rule State** – toggles Decoupling Cap Distance to Via rules on or off.
- **Maximum Distance Between Capacitor and Via** – specifies the maximum distance between the capacitor pin and the via to the power or ground-reference plane.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Maximum Search Distance for Via** – specifies a maximum search distance for a via when a violation is found. This allows you to spot vias that are barely outside the original requirement and therefore not serious violations.
- **Verify Via Connected to Appropriate Plane** – selecting 'no' speeds up EMI Scanner by not verifying the via connects to the selected plane (shape).

- **Power/Ground Trace Decoupling** – displays **Power/Ground-Reference Trace Decoupling** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This rule states that all power and ground-reference traces longer than a specified length must have a decoupling capacitor within a specified distance of the IC power pin.

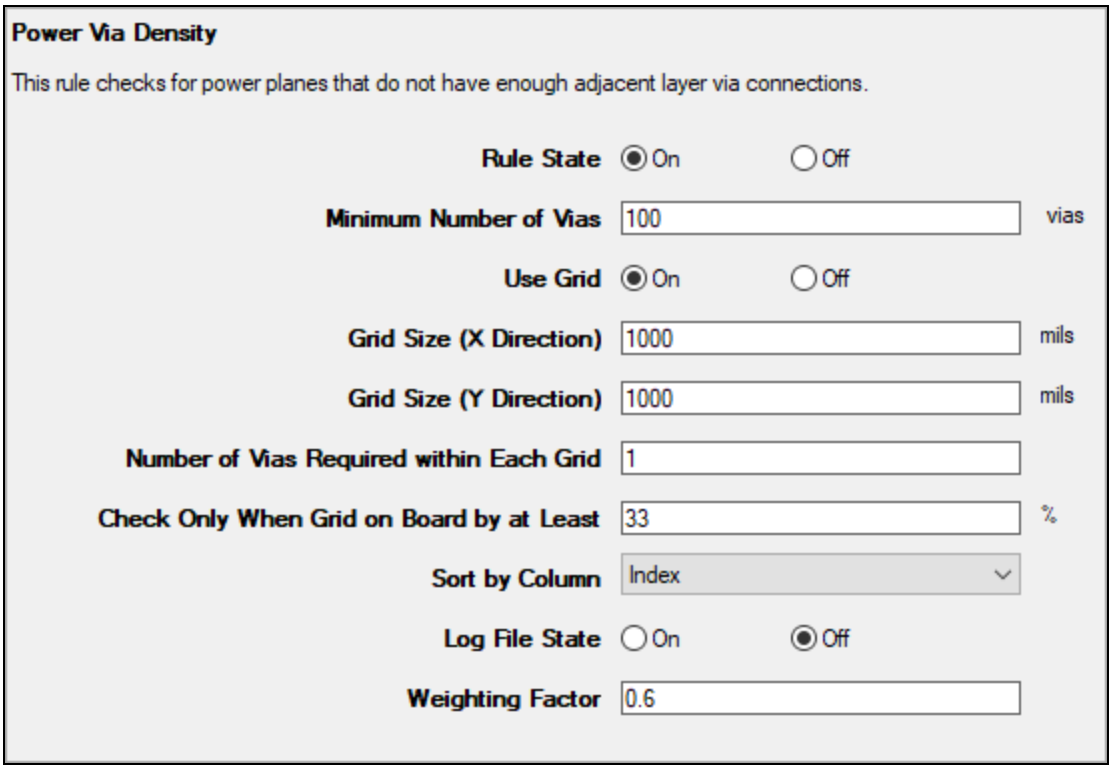
From here, set the following:

- **Rule State** – toggles Power/Ground Trace Decoupling rules on or off.
- **Maximum Ignorable Power/Ground Trace Length** – specifies the maximum power/ground trace length that may be ignored. All traces longer than this length must meet this rule's requirement.
- **Max Distance between Cap and IC Power Pin** – specifies the maximum distance between the capacitor pin and the power or ground-reference pin.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See: Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Power/Via Density** – displays **Power Via Density** rule configuration.



This rule checks for power planes that do not have enough adjacent layer via connections.

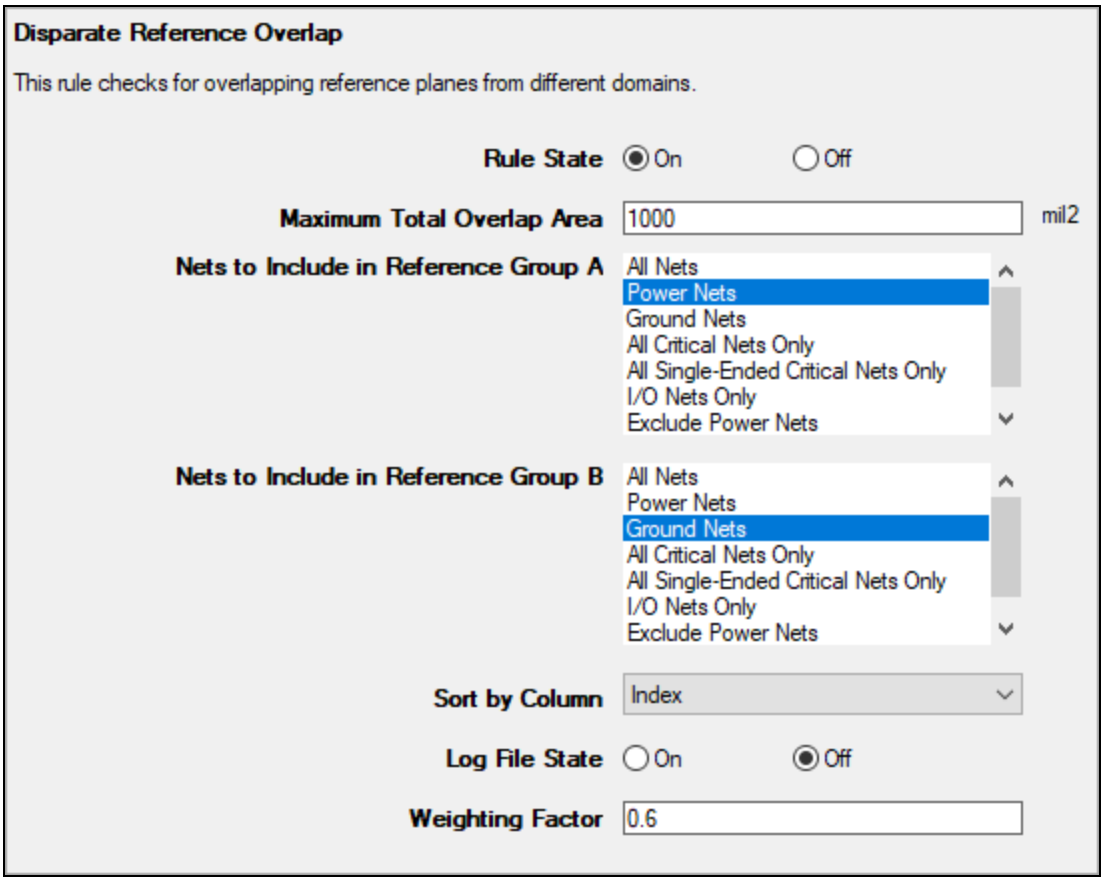
From here, you can set the following:

- **Rule State** – toggles Power Via Density rules on or off.
- **Minimum Number of Vias** – sets the minimum number of vias required to connect adjacent power planes.
- **Use Grid** – toggles the use of a grid on or off.
- **Grid Size (X Direction/Y Direction)** – when Use Grid is enabled, these fields specify the size of the grid.
- **Number of Vias Required within Each Grid** – when Use Grid is enabled, specifies the number of vias required within each grid.
- **Check Only When Grid on Board by at Least** – specifies the percentage of the grid that must be on the board for analysis to occur.
- **Sort by Column** – specifies the column by which results are sorted.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See: Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

- **Reference Overlap** – displays **Disparate Reference Overlap** rule configuration.



The Disparate Reference Overlap rule checks for overlapping reference planes from different domains.

From here, you can set the following:

- **Rule State** – toggles Disparate Reference Overlap on or off.
- **Maximum Total Overlap Area** – specifies the maximum overlap area that is acceptable. Overlap over this maximum will report a violation.
- **Nets to Include in Reference Group A** – selects a net type.
- **Nets to Include in Reference Group B** – selects a net type.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See: Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

>

3. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

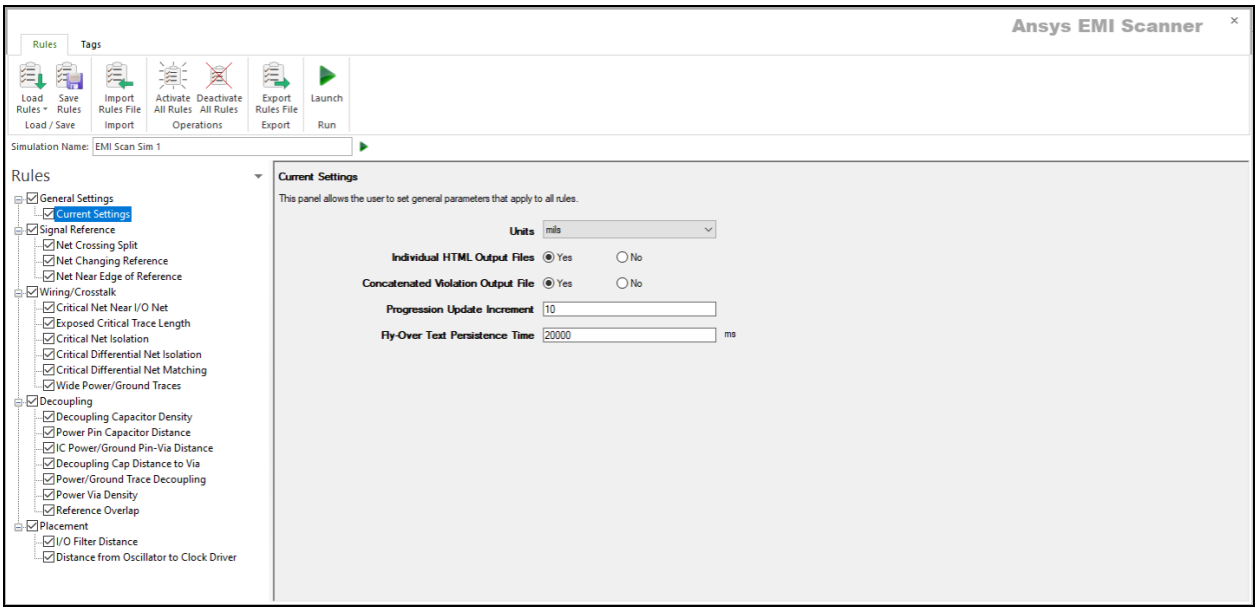
> **Note:**
>
> - You can display a description of a rule parameter as with the cursor over it.
> - Units of measure are set in General Settings.

Applying Placement Rules

To change EMI Scanner Placement rules:

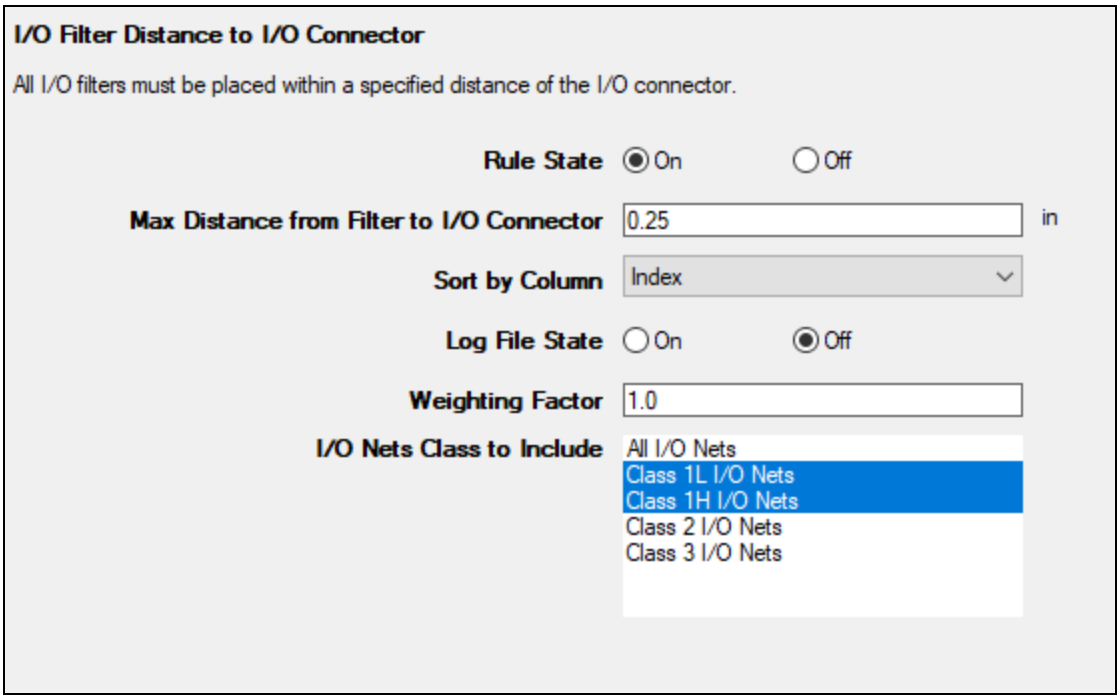
1. Launch EMI Scanner.

   The **EMI Scanner** window opens, on the **Rules** 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. From the Rules pane, select one of the two subcategories under **Placement**:
   - **I/O Filter Distance** – displays the **I/O Filter Distance to I/O Connector** rule configuration.



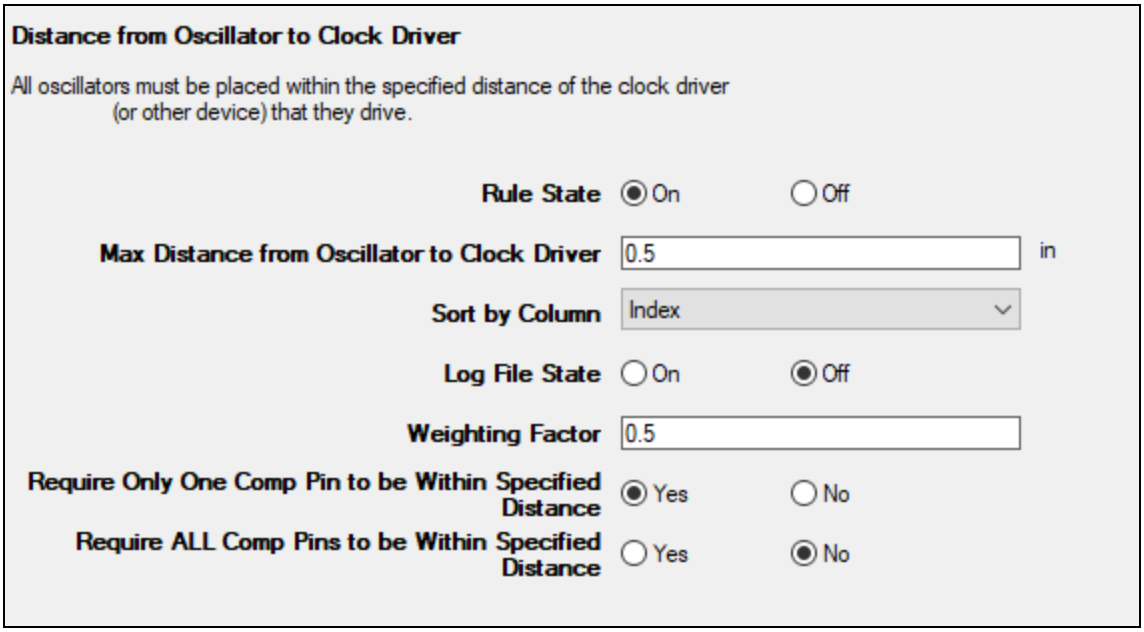
This rule states that all I/O filters must be placed within a specified distance of the I/O connector.

From here, set the following:

- **Rule State** – toggles I/O Filter Distance rules on or off.
- **Max Distance from Filter to I/O Connector** – specifies the maximum distance a filter may be placed on the I/O connector.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **I/O Nets to Include** – specifies the type(s) of I/O Nets to include. Select one, or **Ctrl**+click to select multiple types. See Technical Note.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Distance from Oscillator to Clock Driver** – displays the **Distance from Oscillator to Clock Driver** rule configuration.



This rule states that all oscillators must be placed within a specified distance on the clock driver or other device they drive.

From here, set the following:

- **Rule State** – toggles Distance from Oscillator to Clock Driver rules on or off.
- **Max Distance from Oscillator to Clock Driver** – specifies the distance within which all oscillators must be placed of the clock driver (or other device) they drive.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Require Only One Comp Pin to be Within Specified Distance** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Require ALL Comp Pins to be Within Specified Distance** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.

3. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

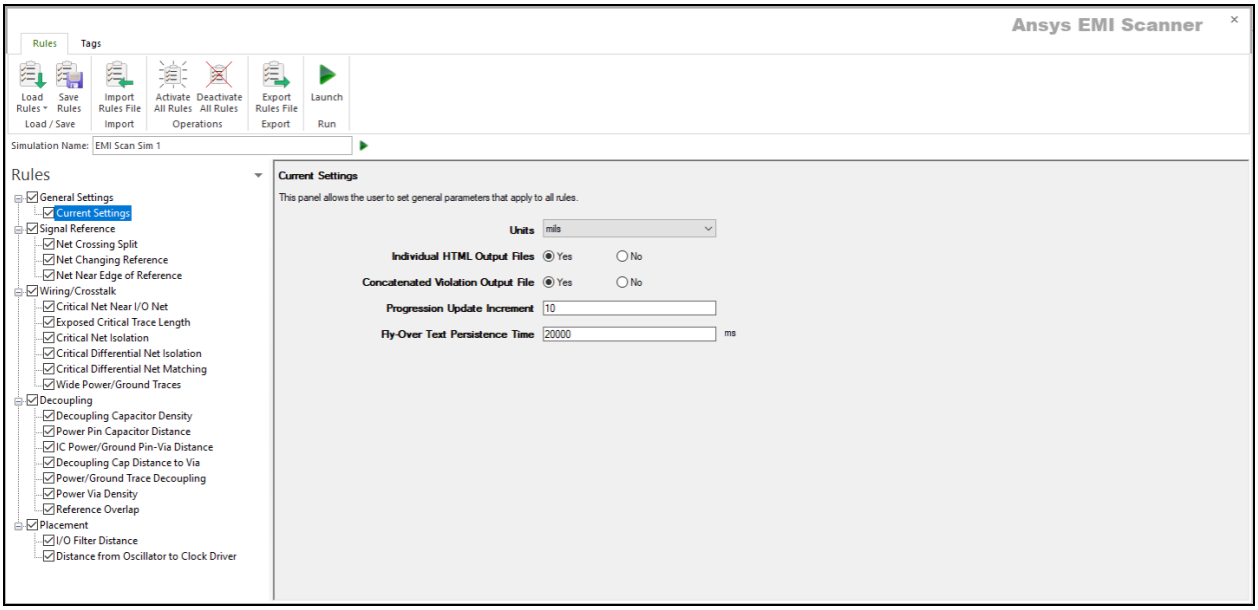
> **Note:**
>
> - You can display a description of a rule parameter at any time as with the cursor over it.
> - Units of measure are set in General Settings.

Applying Net Integrity Rules

To change EMI Scanner Net Integrity rules:

1. Launch EMI Scanner.

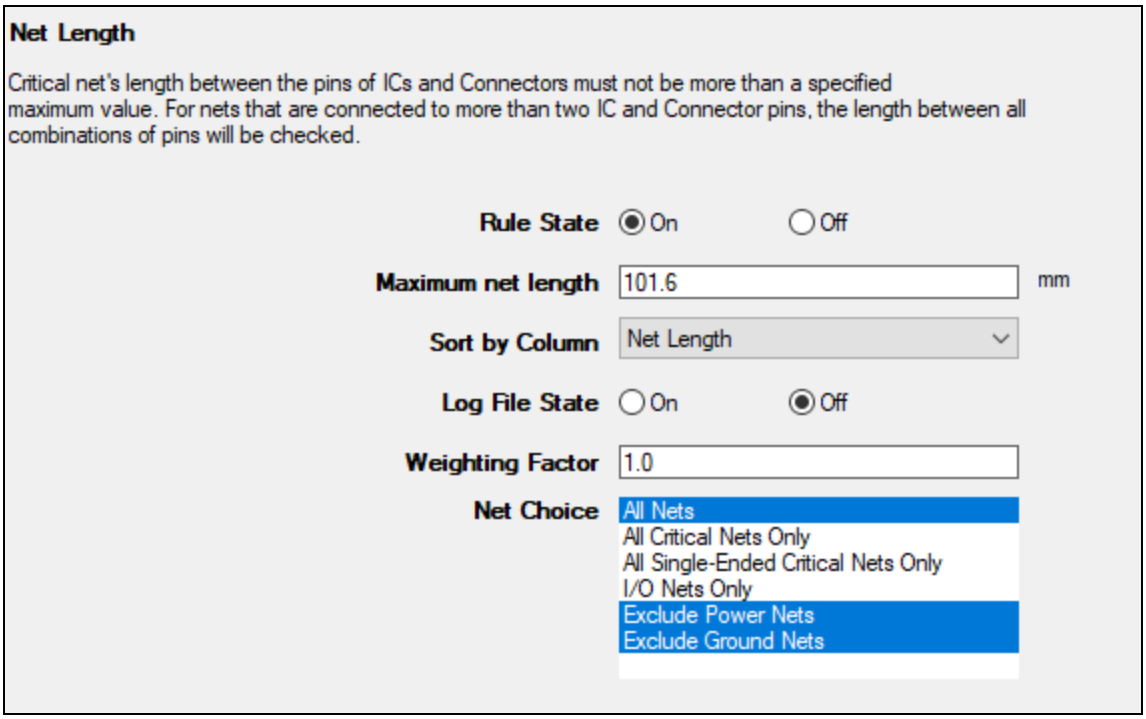
   The **EMI Scanner** window opens, on the **Rules** tab.



2. Click **Load Rules** and select a profile containing Signal Integrity rules (e.g., Default SI Rules Profile).

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 Rules pane, select one of the eight subcategories under **Net Integrity**:
   - **Net Length** – displays the **Net Length** rule configuration.



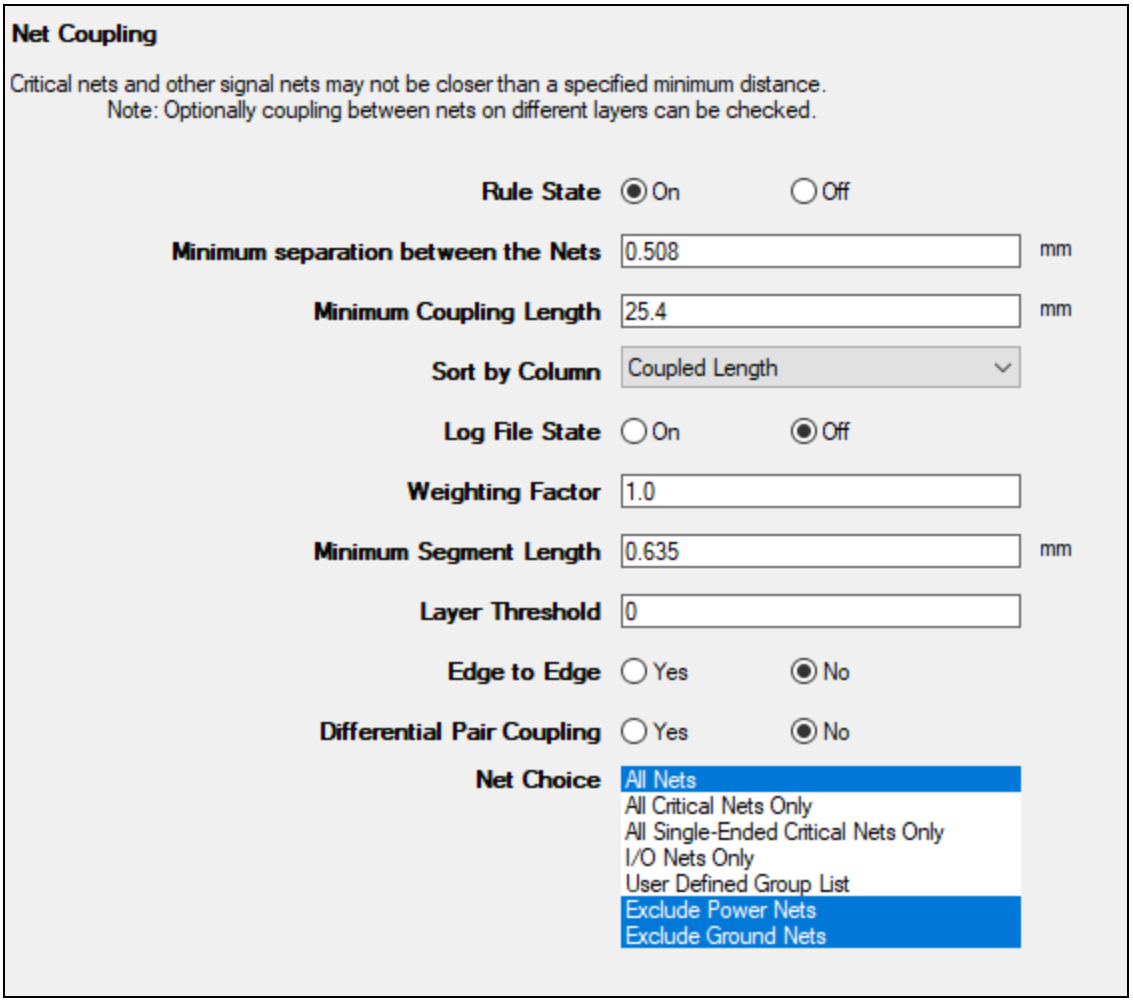
This rule states that a critical net's length between pins of ICs and Connectors must not be more than a specified value.

From here, you can set the following:

- **Rule State** – toggles Net Length rules on or off.
- **Maximum Net Length** – specifies the maximum net length.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Net to Net Coupling** – displays the **Net Coupling** rule configuration.



This rule states that critical nets and other signal nets may not be closer than a specified minimum distance.

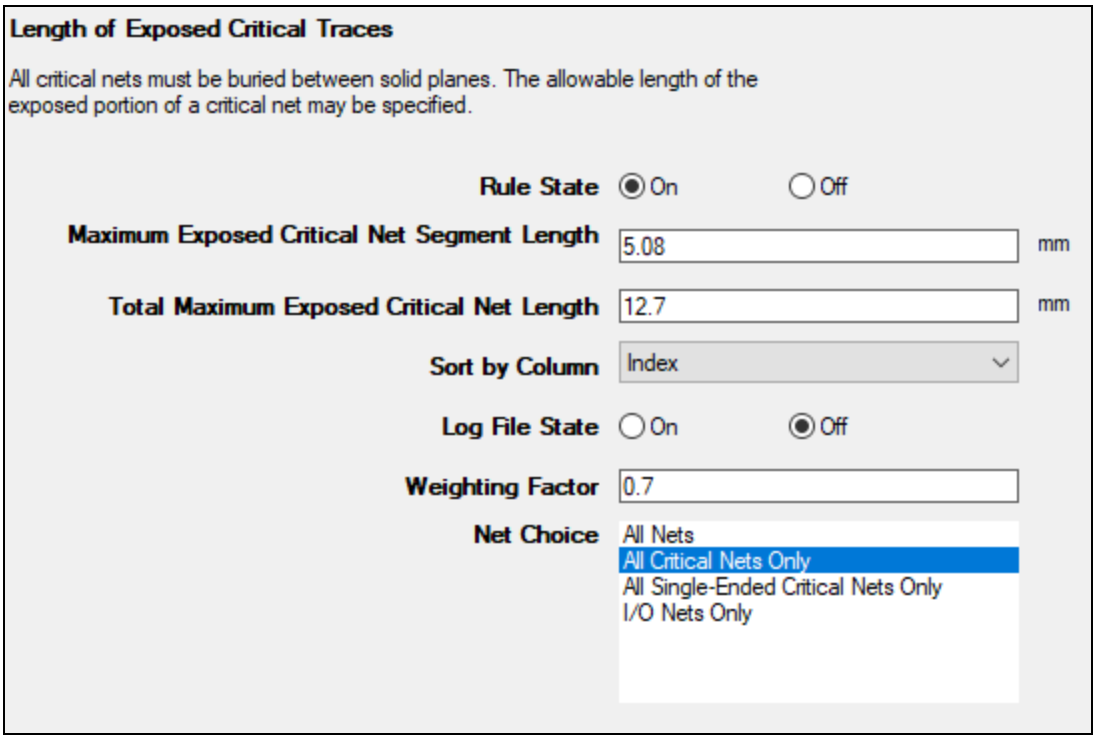
From here, you can set the following:

- **Rule State** – toggles Net Coupling rules on or off.
- **Minimum Separation Between the Nets** – controls how close the nets can be before they are considered coupled.
- **Maximum Coupling Length** – specifies how much coupling must exist before a violation is reported.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Minimum Segment Length** – specifies the smallest net segment length to check for coupling. Any segments smaller than this length will be ignored.
- **Layer Threshold** – sets the number of layers above and below to check for coupling (e.g., 0 = only the selected layer, 1 = the selected layer plus one layer above and one layer below).
- **Edge to Edge** – by default, the minimum distance between nets is based on center-to-center measurement. Selecting this option changes the measurement to edge-to-edge.
- **Differential Pair Coupling** – determines whether to allow or ignore coupling between the nets that make up a differential pair.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Exposed Critical Trace Length** – displays the **Length of Exposed Critical Traces** rule configuration.



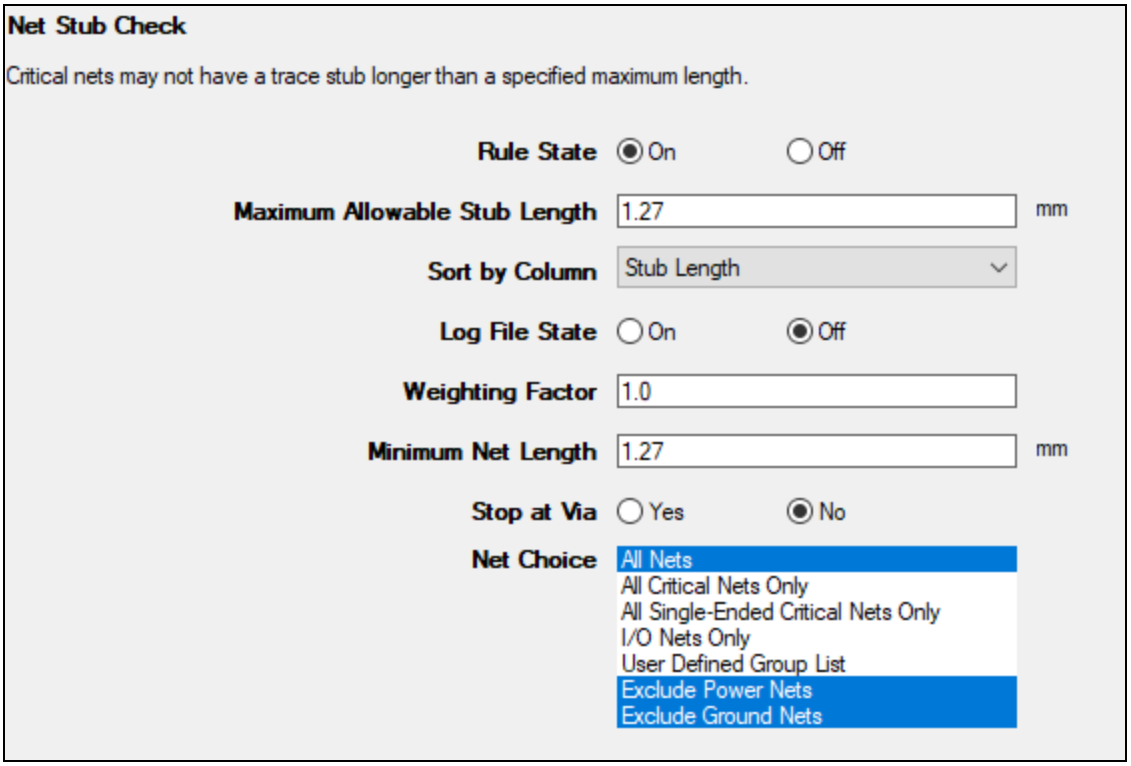
This rule states that all critical nets must be buried between solid planes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From here, you can set the following:

- **Rule State** – toggles Net Length rules on or off.
- **Maximum Exposed Critical Net Segment Length** – specifies the maximum critical net segment length allowed on external (exposed) layers.
- **Total Maximum Exposed Critical Net Length** – specifies the total maximum critical net length allowed on external (exposed) layers (e.g., multiple segments on a critical net may be exposed, but short enough to not cause a violation individually).
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Net Stub** – displays the **Net Stub Check** rule configuration.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

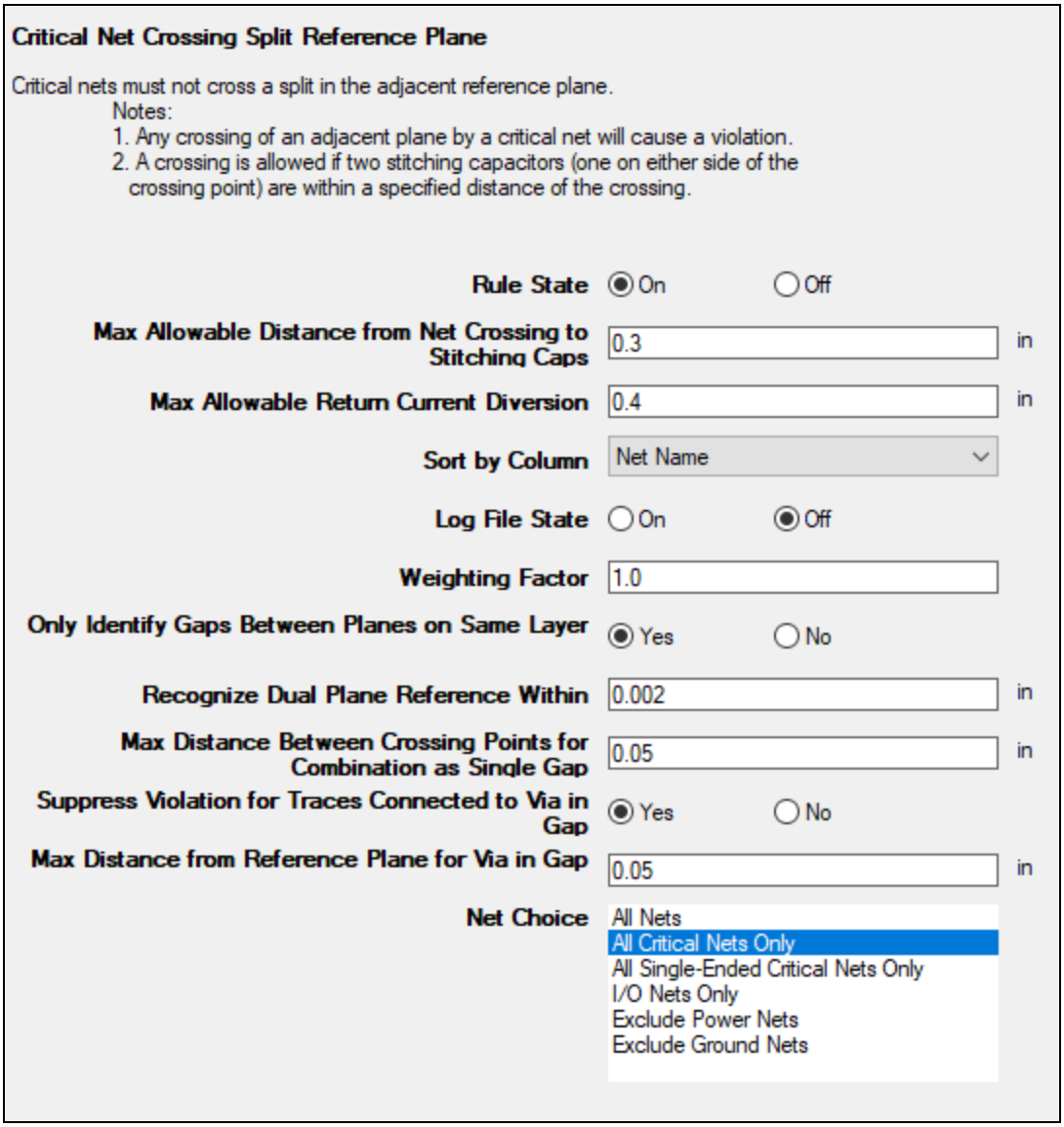
This rule states that critical nets may not have a trace stub longer than the specified maximum length.

From here, you can set the following:

- **Rule State** – toggles Net Stub Check rules on or off.
- **Maximum Allowable Stub Length** – specifies the maximum stub length. Stubs longer than this are violations.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Minimum Net Length** – sets the minimum length at which nets are included.
- **Stop at Via** – determines whether vias count as a stub termination condition. By defauly, the termination of a stub is a pin or a fork in the net, whichever is found first. Setting this parameter to 'yes' adds vias (to pins and forks) as a stub termination condition.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Net Crossing Split** – displays the **Critical Net Crossing Split Reference Plane** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This rule states that critical nets must not cross a split in the adjacent reference plane.

From here, you can set the following:

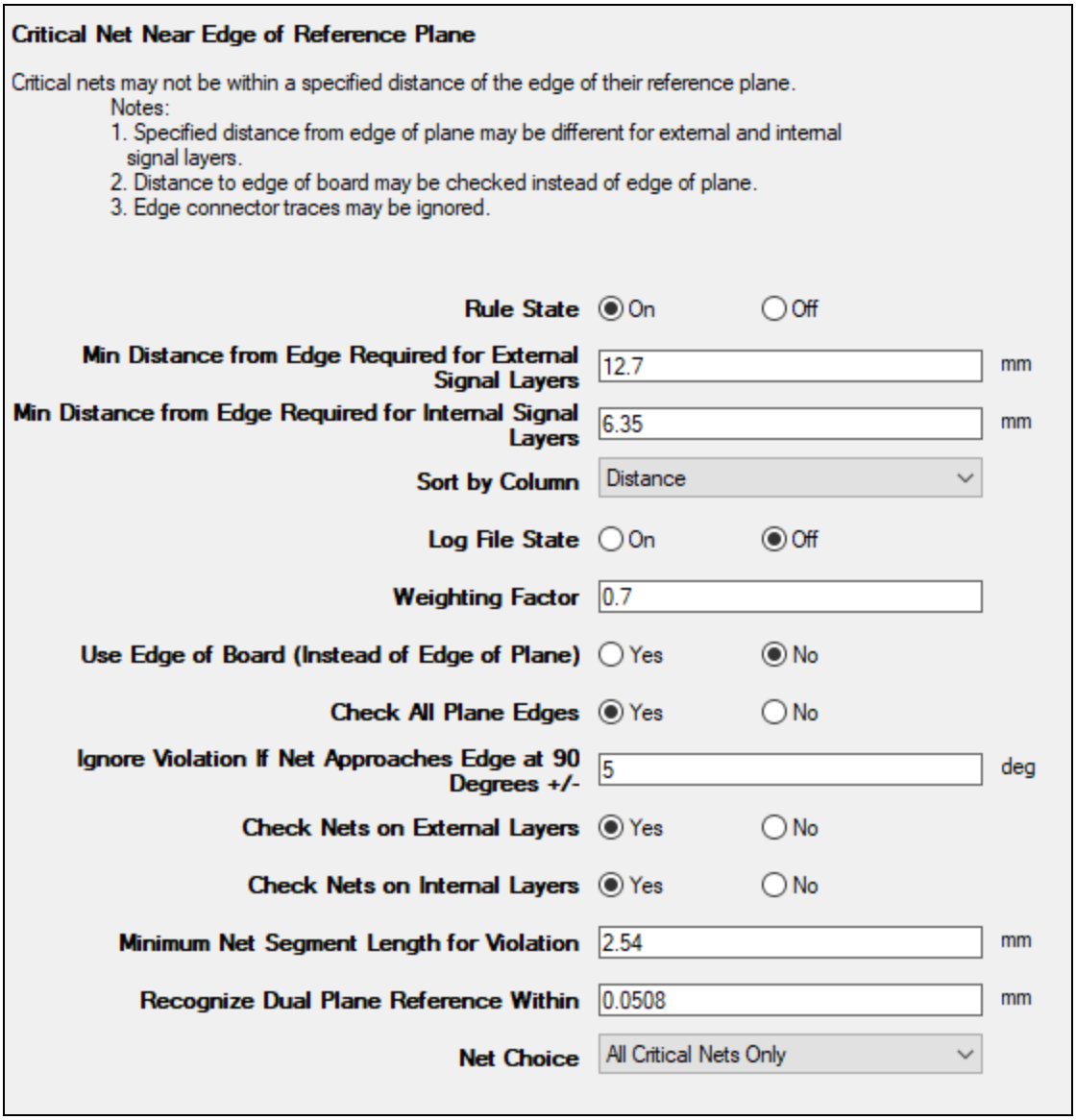
- **Rule State** – toggles Net Crossing Split rules on or off.
- **Max Allowable Distance from Net Crossing to Stitching Caps** – sets the distance on the point where the net crosses a split in its reference plane at which two stitching capacitors must be found.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Max Allowable Return Current Diversion** – determines whether a signal's return current can divert itself around the gap in the reference plane. The total distance of the diversion must be less than this value.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Only Identify Gaps Between Planes on Same Layer** – chooses whether gaps are only between planes on the same layer. When the rule tries to pair up nearby plane boundary crossing points and identify gaps, this parameter will enable or disable a check for layer commonality.
- **Recognize Dual Plane Reference Within** – specifies a distance and instructs EMI Scanner to consider both planes as a reference when checking for gap crossing when the distance difference to each plane is less than the specified distance. This allows recognition of both planes as references for strip line traces when the trace is offset on the exact center by a small amount (normally for manufacturing reasons).
- **Max Distance Between Crossing Points for Combination as a Single Gap** – specifies the distance under which nearby crossing points are combined into a single violation. Setting too large a value can cause the algorithm to pair up unrelated crossing points, and setting too small a value can prevent crossing points from being paired up, resulting in more violations. It is recommended to set the value slightly larger than the typical plane separation within a layer, based on the board design technology being used.
- **Suppress Violation for Traces Connected to Via in Gap** – instructs EMI Scanner whether to ignore trace crossing split/gap when connected to a via within that gap.
- **Max Distance from Reference Plane for Via in Gap** – specifies the maximum distance from edge of the reference plane to the via. Violations will not be suppressed (from above) for vias within gap if the trace extends greater than the specified distance.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Net Near Edge of Reference** – displays the **Critical Net Near Edge of Reference Plane** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This rule states that critical nets may not be within a specified distance of the edge of their reference plane.

From here, you can set the following:

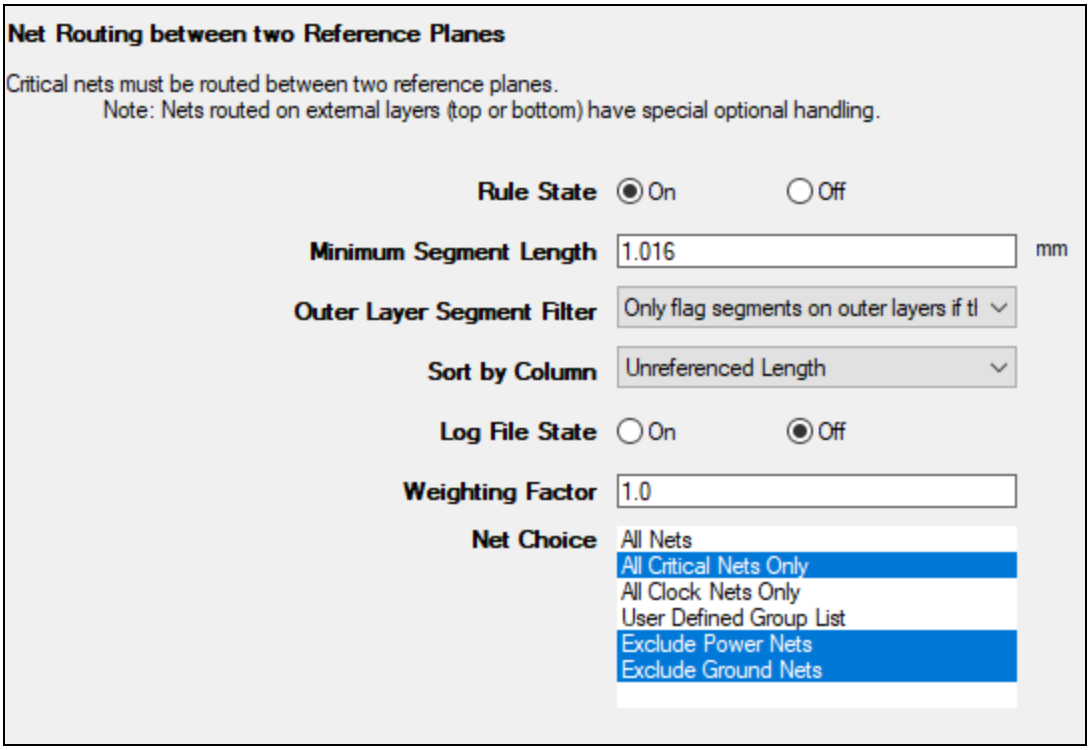
- **Rule State** – toggles Net Near Edge of Reference rules on or off.
- **Min Distance from Edge Required for External Signal Layers** – specifies the minimum distance that must exist between the edge of the reference plane and a critical signal net when the net is on an external layer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Min Distance from Edge Required for Internal Signal Layers** – specifies the minimum distance that must exist between the edge of the reference plane and a critical signal net when the net is on an internal layer.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Use Edge of Board (Instead of Edge of Plane)** – allows EMI Scanner to check distance from critical nets to the edge of the board instead of the edge of the reference plane. This can be useful if the board is only partially complete and all reference planes are not yet completed.
- **Check All Plane Edges** – checks the distance from nets to any edge of their reference plane, rather than limiting the check to reference plane edges near the edge of the board.
- **Ignore Violation If Net Approaches Edge at 90 Degrees +/-** – EMI scanner ignores edge connectors that approach the edge of the reference plane or board at a 90 degree angle, tolerating approaches slightly off a perfect 90 degrees.
- **Check Nets on External Layers** – specifies whether nets on external layers should be checked.
- **Check Nets on Internal Layers** – specifies whether nets on internal layers should be checked.
- **Minimum Net Segment Length for Violation** – specifies the minimum net segment length before EMI Scanner will report a violation if the net is in the specified distance on the edge of the reference plane (or board).
- **Recognize Dual Plane Reference Within** – specifies a distance and instructs EMI Scanner to consider both planes as a reference when checking for traces too close to the edge of reference planes when the distance difference to each plane is less than the specified distance. This allows recognition of both planes as references for strip line traces when the trace is offset on the exact center by a small amount (normally for manufacturing reasons).
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Between Ref Plane Routing** – displays the **Net Routing Between Two Reference Planes** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



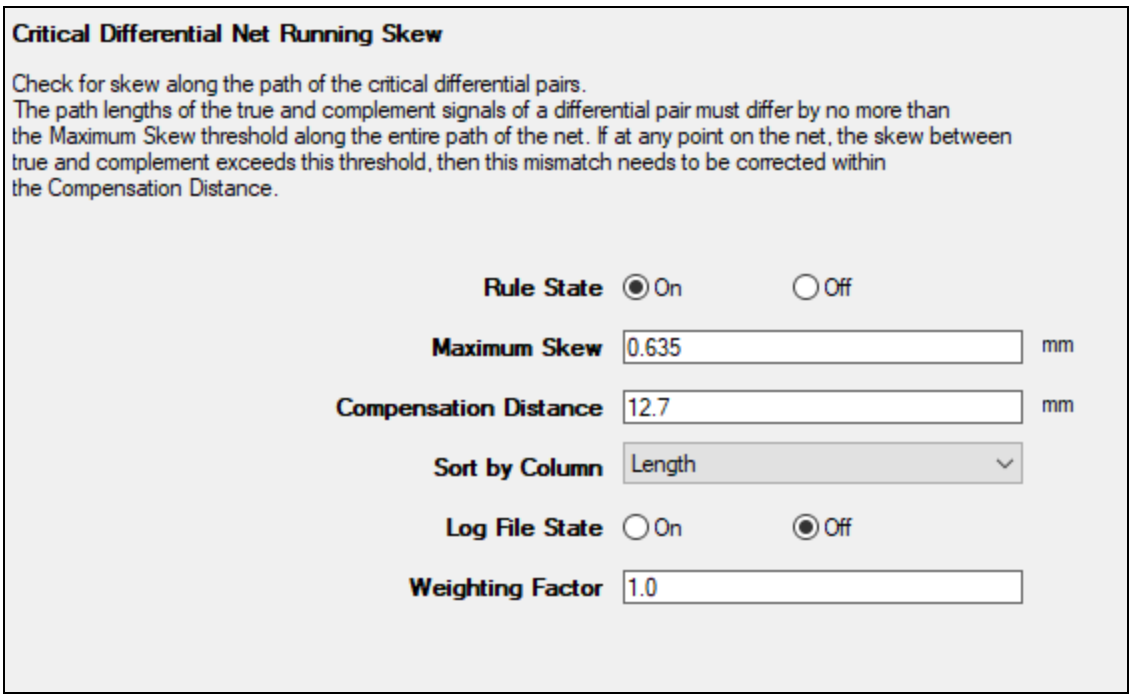
This rule states that critical nets must be routed between two reference planes.

From here, you can set the following:

- **Rule State** – toggles Net Routing Between Two Reference Planes rules on or off.
- **Minimum Segment Length** – specifies the minimum net segment length before EMI Scanner will report a violation.
- **Outer Layer Segment Filter** – specifies how segments on outer layers are treated.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Diff Running Skew** – displays the **Critical Differential Net Running Skew** rule configuration.



This rule checks for skew along the path of critical differential pairs.

From here, you can set the following:

- **Rule State** – toggles Diff Running Skew rules on or off.
- **Maximum Skew** – sets the maximum allowable skew.
- **Compensation Distance** – skew that exceeds the maximum allowable skew must be corrected within this distance.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

4. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

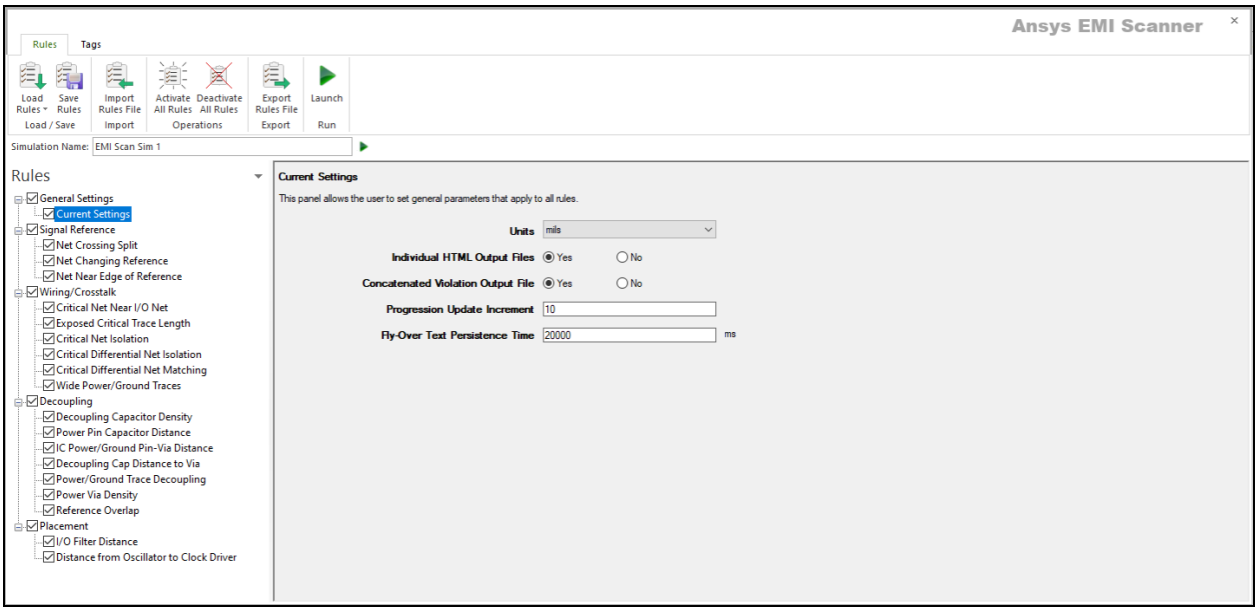
- You can display a description of a rule parameter at any time by hovering the cursor over it.
- Units of measure are set in General Settings.

Applying Via Integrity Rules

To change EMI Scanner Via Integrity rules:

1. Launch EMI Scanner.

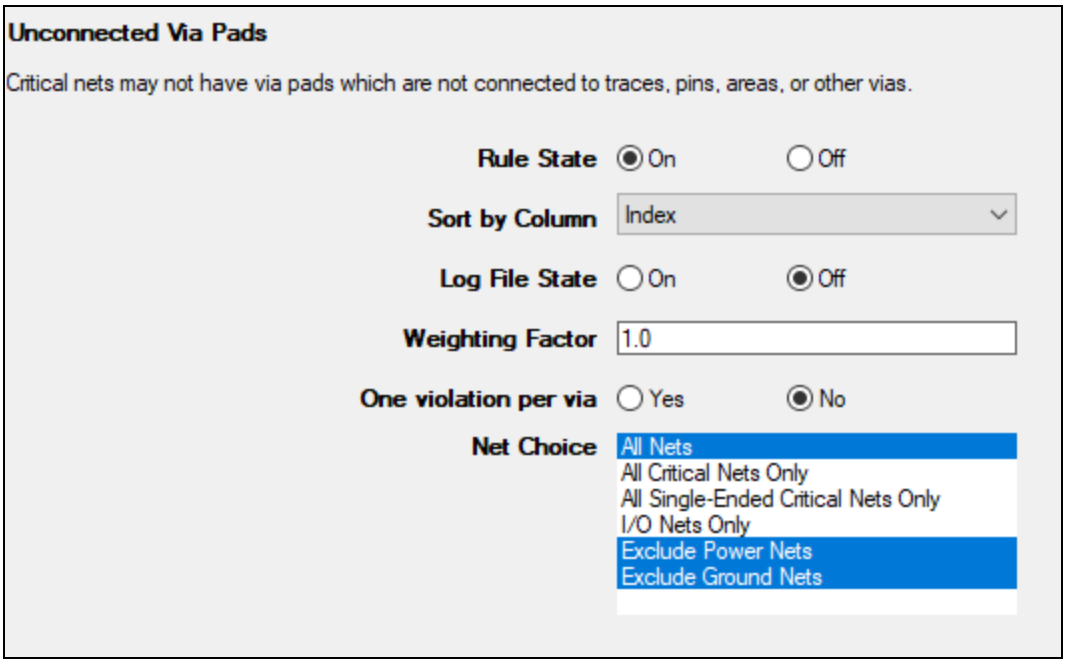
   The **EMI Scanner** window opens, on the **Rules** tab.



2. Select a profile containing Via Integrity rules (e.g., Default SI Rules Profile) on the **Load Rules** 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.

3. From the Rules pane, select one of the four subcategories under **Via Integrity**:
   - **Unconnected Via Pads** – displays the **Unconnected Via Pads** rule configuration.



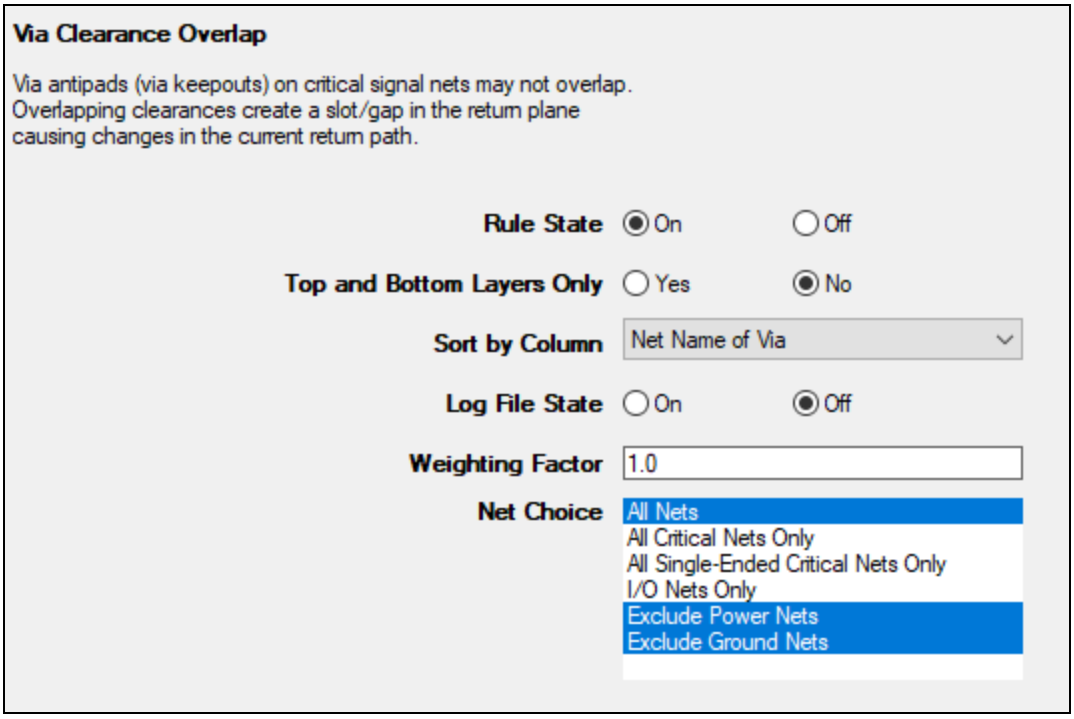
This rule states that critical nets may not have via pads that are not connected to traces, pins, areas, or other vias.

From here, set the following:

- **Rule State** – toggles Unconnected Via Pads rules on or off.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **One Violation Per Via** – when **Yes** is selected, multiple violations on a via are counted as one violation.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Via Clearance Overlap** – displays the **Via Clearance Overlap** rule configuration.



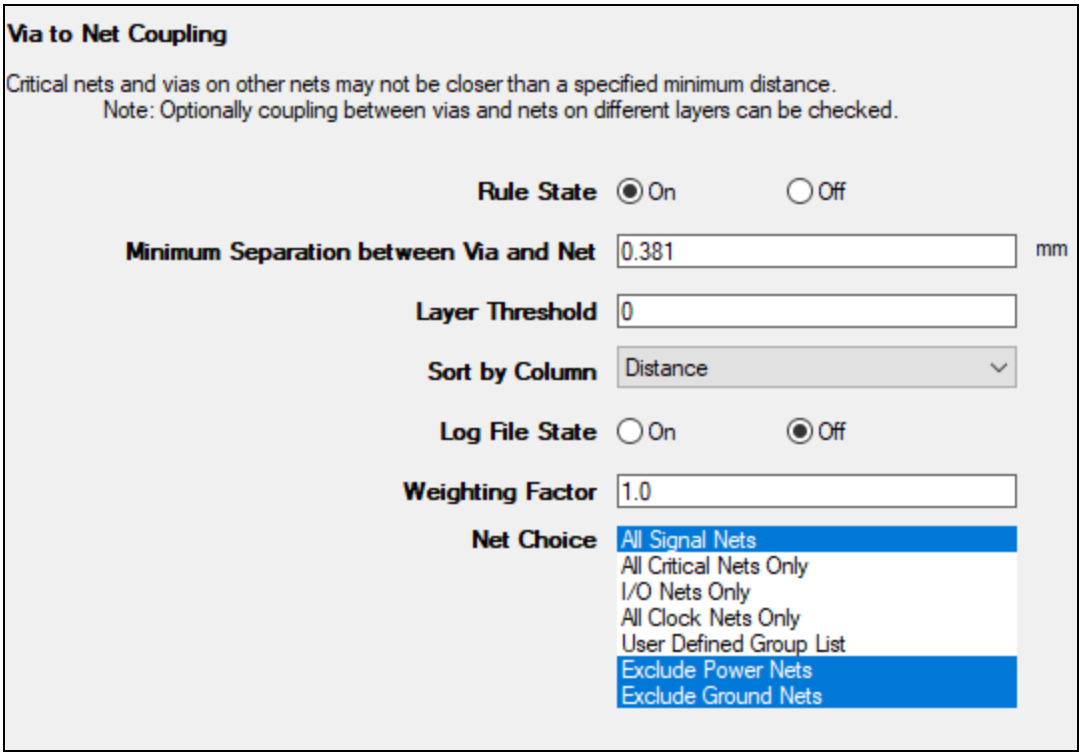
This rule states that via antipads on critical signal nets may not overlap.

From here, set the following:

- **Rule State** – toggles Via Clearance Overlap rules on or off.
- **Top and Bottom Layers Only** – limits analysis to the top and bottom layers only.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Via to Net Coupling** – displays the **Via to Net Coupling** rule configuration.



This rule states that critical nets and vias on other nets may not be closer than a specified minimum distance.

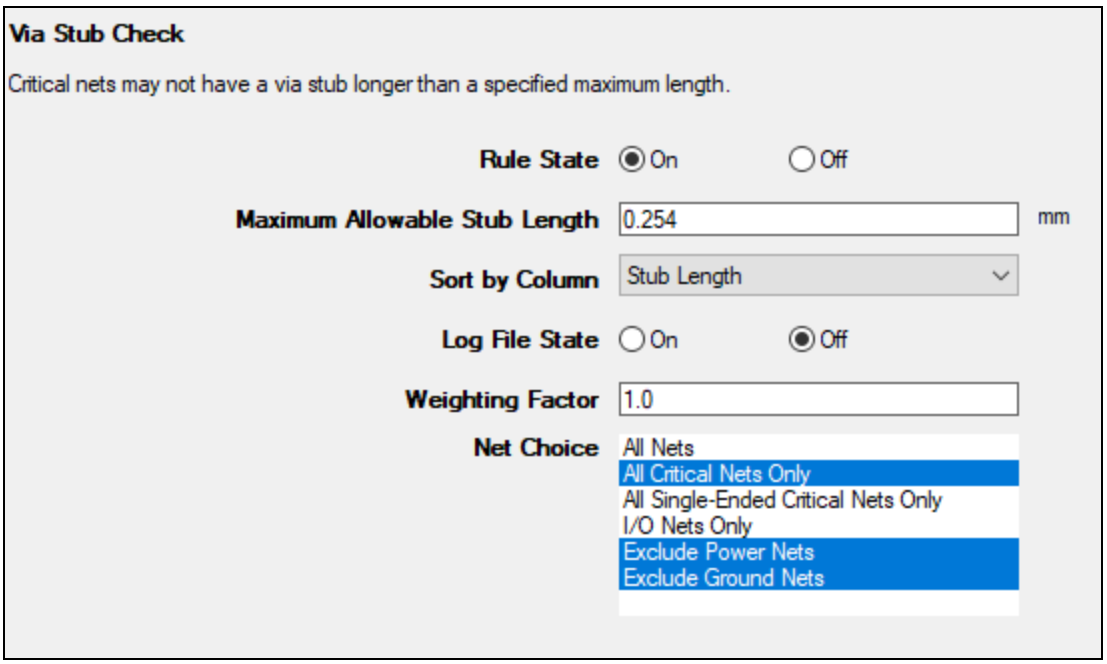
From here, set the following:

- **Rule State** – toggles Via to Net Coupling rules on or off.
- **Minimum Separation Between Via and Net** – controls how close the net and the via can be before they are considered coupled.
- **Layer Threshold** – sets the number of layers above and below to check for coupling (e.g., 0 = only the selected layer, 1 = the selected layer plus one layer above and one layer below).
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Via Stub** – displays the **Via Stub Check** rule configuration.



This rule states that critical nets may not have a via stub longer than a specified maximum length.

From here, set the following:

- **Rule State** – toggles Via Stub rules on or off.
- **Maximum Allowable Stub Length** – specifies the maximum stub length. Any stub longer than this setting is a violation.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

Note:

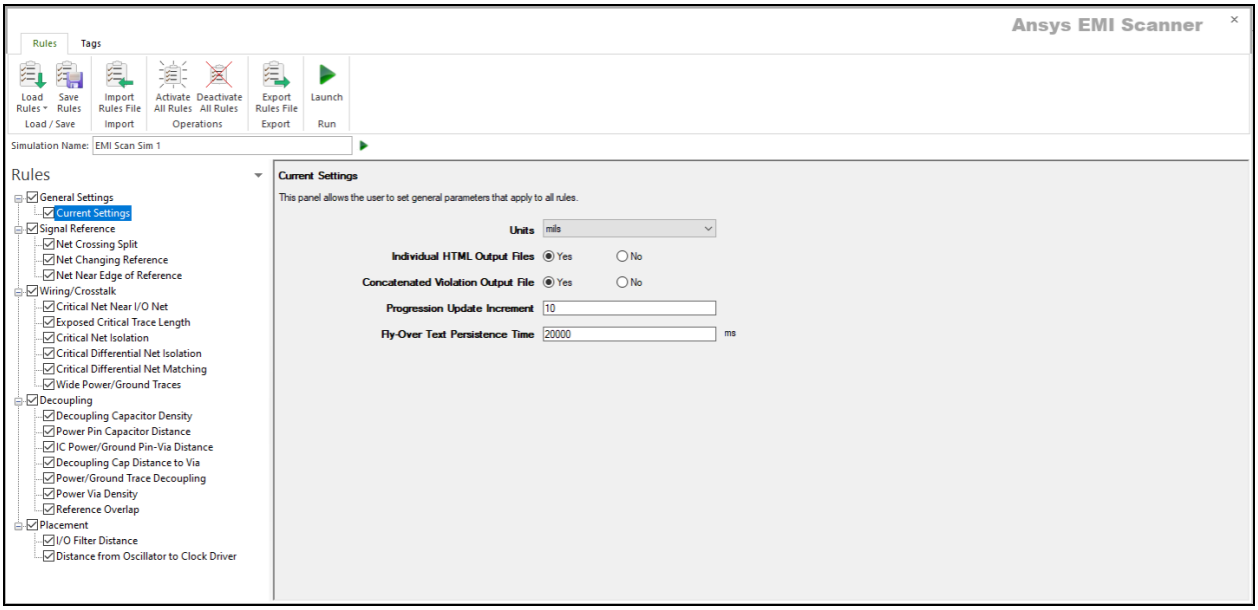
- You can display a description of a rule parameter at any time as with the cursor over it.
- Units of measure are set in General Settings.

Applying Power Integrity Rules

To change EMI Scanner Power Integrity rules:

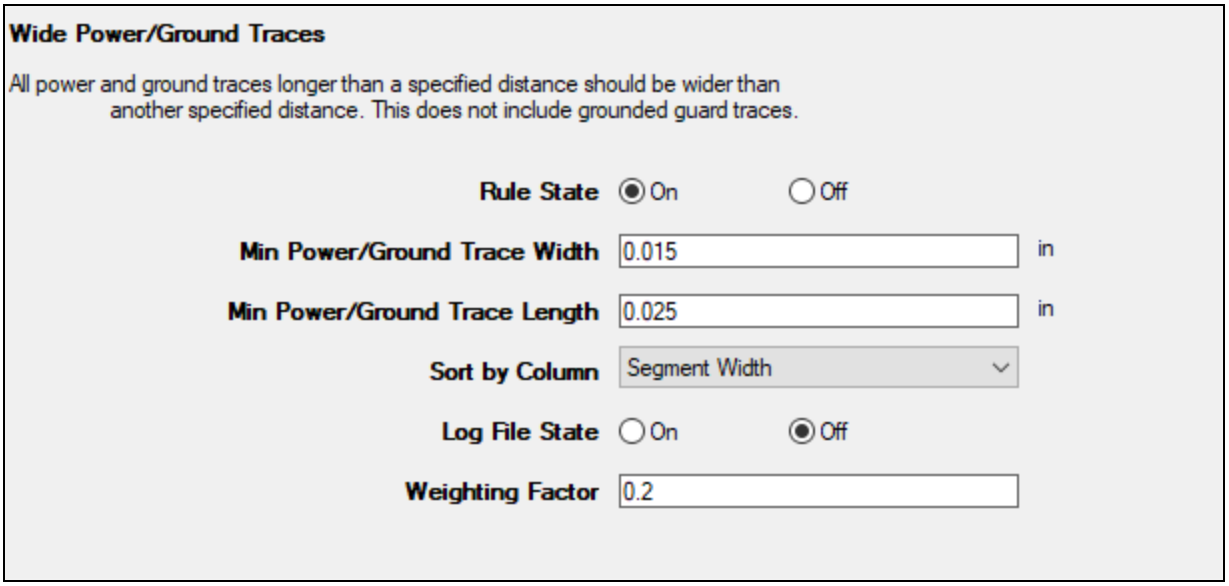
1. Launch EMI Scanner.

   The **EMI Scanner** window opens, on the **Rules** tab.



2. Click **Load Rules** and select a profile containing Power Integrity rules (e.g., Default SI Rules Profile).
3. From the Rules pane, select one of the seven subcategories under **Power Integrity**:
   - **Wide Power/Ground Traces** – displays the **Wide Power/Ground Traces** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



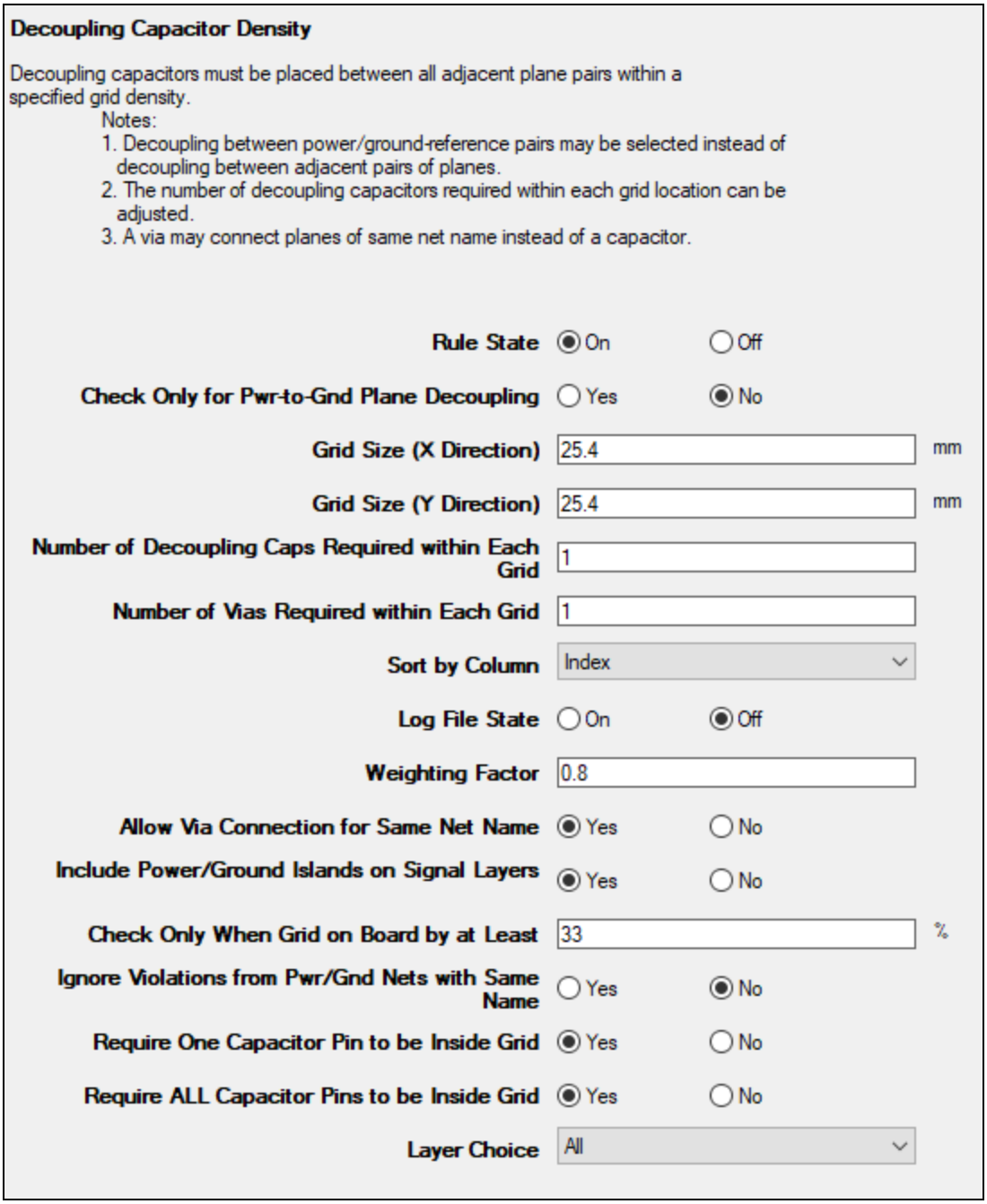
This rule states that all power and ground traces longer than a specified distance should be wider than another specified distance.

From here, set the following:

- **Rule State** – toggles Wide Power/Ground Traces rules on or off.
- **Min Power/Ground Trace Width** – sets the minimum width limit for all power/ground traces (except grounded guard traces).
- **Min Power/Ground Trace Length** – sets the minimum length limit for all power/ground traces (except grounded guard traces).
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

- **Decoupling Capacitor Density** – displays the **Decoupling Capacitor Density** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This rule states that decoupling capacitors must be placed between all adjacent plane pairs within a specified grid density.

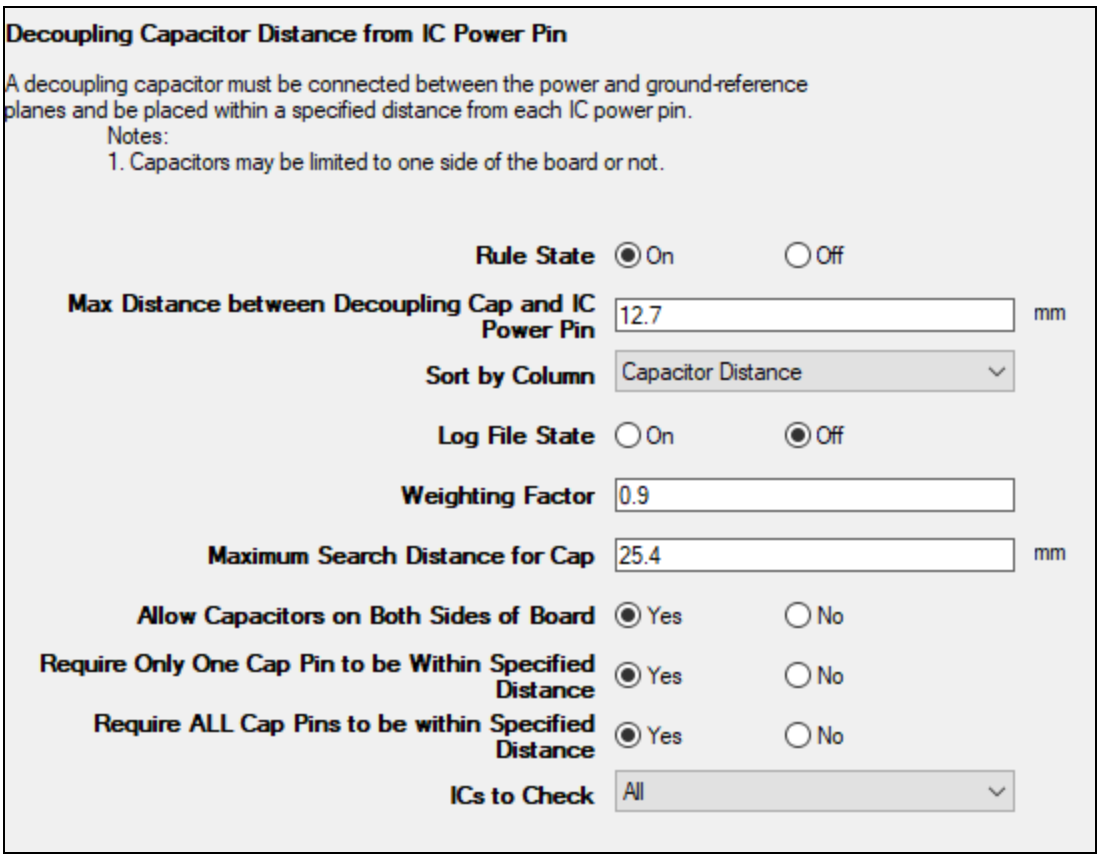
From here, set the following:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Rule State** – toggles Decoupling Capacitor Density rules on or off.
- **Check Only for Pwr-to-Gnd Plane Decoupling** – sets whether to ignore plane-to-plane decoupling and check only for decoupling capacitors between power and ground planes.
- **Grid Size (X Direction)** – specifies the size of the grid used to check for decoupling capacitors. Used in conjunction with **Grid Size (Y Direction)**.
- **Grid Size (Y Direction)** – specifies the size of the grid used to check for decoupling capacitors. Used in conjunction with **Grid Size (X Direction)**.
- **Number of Decoupling Caps Required within Each Grid** – specifies the number of decoupling capacitors connecting between planes that are required within each grid location.
- **Number of Vias Required within Each Grid** – specifies the number of vias connecting between planes that are required within each grid location. You must select **Allow Via Connection for Same Net Name** to use this option.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Allow Via Connection for Same Net Name** – determines whether EMI Scanner allows a via in place of a decoupling capacitor when the two planes are the same net name.
- **Include Power/Ground Islands on Signal Layers** – specifies whether to check for decoupling on power/ground islands.
- **Check Only When Grid on Board by at Least** – specifies the percentage of a grid that must be on the board in order for it to be checked.
- **Ignore Violations from Pwr/Gnd Nets with Same Name** – determines that only one plane of a given name is checked for violations (e.g., if two planes are "gnd" and a violation occurs between gnd and another layer, only one violation is reported).
- **Require One Capacitor Pin to be Inside Grid** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **Require ALL Capacitor Pins to be Inside Grid** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **Layer Choice** – specifies the layers to include in this rule analysis.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Power Pin Capacitor Distance** – displays the **Decoupling Capacitor Distance from IC Power Pin** rule configuration.



This rule states that a decoupling capacitor must be connected between the power and ground-reference planes and be placed within a specified distance of each IC power pin.

From here, set the following:

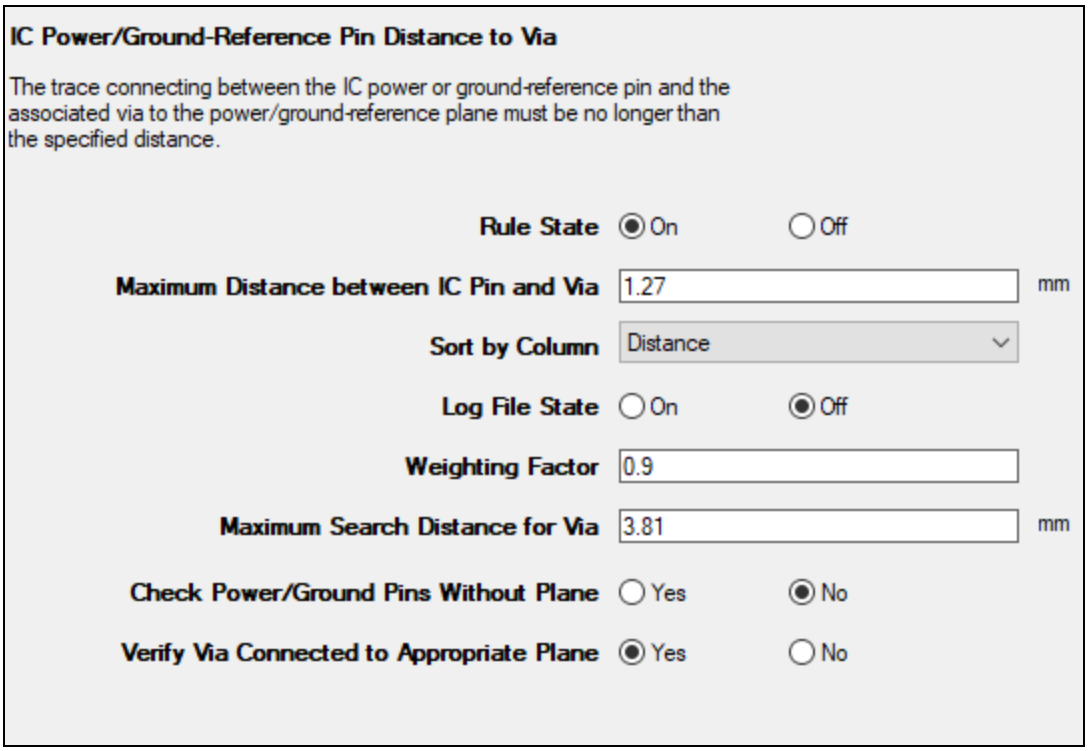
- **Rule State** – toggles Power Pin Capacitor Distance rules on or off.
- **Max Distance Between Decoupling Cap and IC Power Pin** – specifies the maximum distance between an IC power pin and its decoupling capacitor.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

- **Maximum Search Distance for Cap** – specifies the maximum search distance for a capacitor. A larger search box increases the chances of EMI Scanner finding capacitors, but can slow processing time.
- **Allow Capacitors on Both Sides of Board** – allows EMI Scanner to include capacitors on both sides of the board to satisfy this rule.
- **Require Only One Cap Pin to be within Specified Distance** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **Require ALL Cap Pins to be within Specified Distance** – changes how EMI Scanner determines whether a component is inside or outside the grid. See Technical Note.
- **ICs to Check** – specifies the types of components to include in this rule analysis.

- **IC Power/Ground Pin-Via Distance** – displays the **IC Power/Ground-Reference Pin Distance to Via** rule configuration.



This rule states the trace connecting the IC power or ground-reference pin and the associated via to the ground/reference plane must be no longer than a specified

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

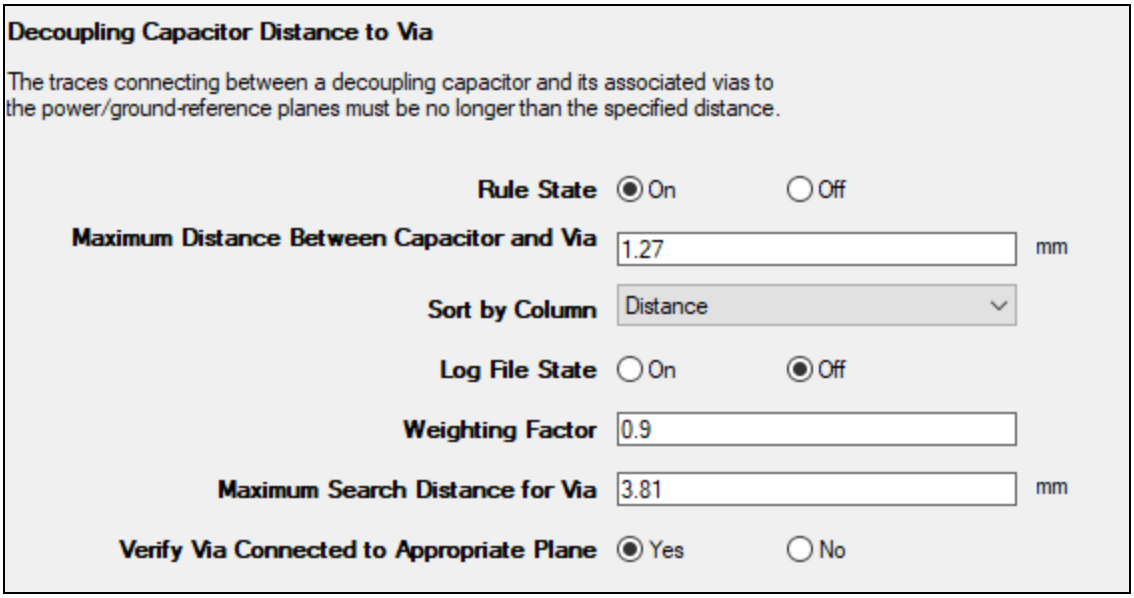
distance..

From here, set the following:

- **Rule State** – toggles IC Power/Ground Pin-Via Distance rules on or off.
- **Maximum Distance Between IC Pin and Via** – specifies the maximum distance between the IC power or ground-reference pin and the via to the power or ground-reference plane.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Maximum Search Distance for Via** – specifies the maximum search distance for a via when a violation is found. This allows users to quickly determine if the via is barely outside the original requirement and therefore not a serious violation.
- **Check Power/Ground Pins Without Plane** – specifies whether to check the distance between the IC power/ground pin and its via when the power/ground plane has not been completed yet.
- **Verify Via Connected to Appropriate Plane** – specifies whether EMI Scanner verifies the via connects to the selected plane (shape). Selecting 'no' speeds up processing.

- **Decoupling Cap Distance to Via** – displays the **Decoupling Capacitor Distance to Via** rule configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



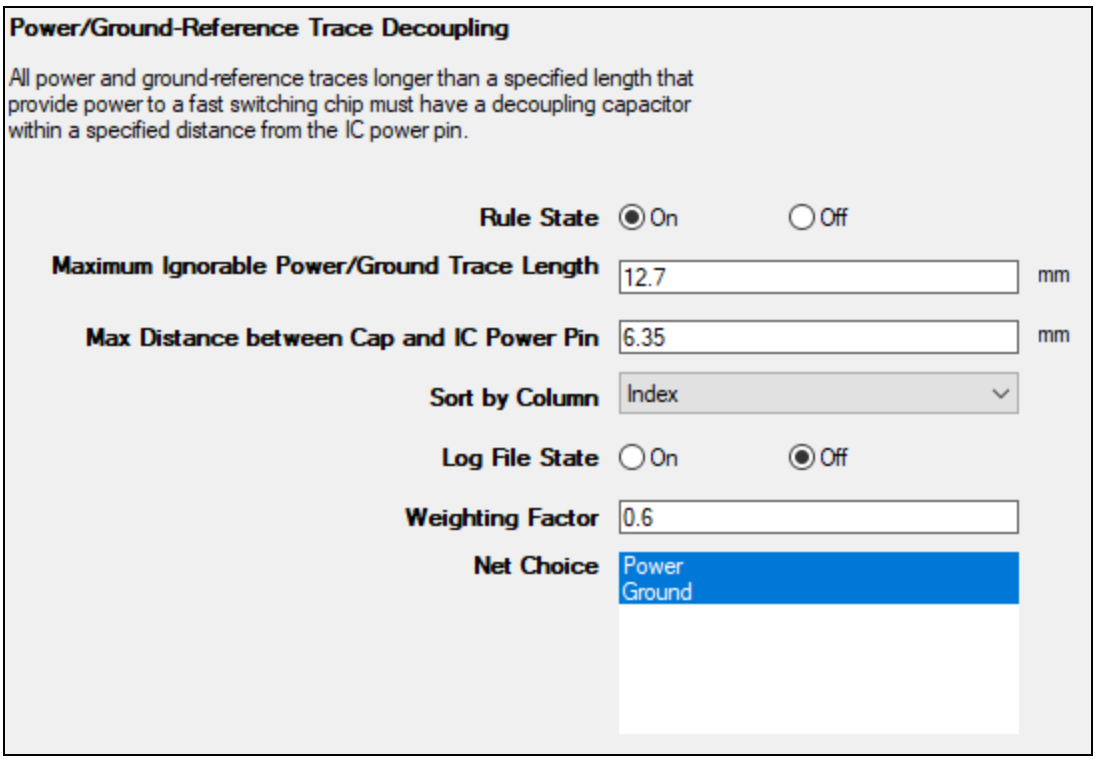
This rule states the traces connecting a decoupling capacitor and its associated vias to the power/ground-reference planes must be no longer than a specified distance.

From here, set the following:

- **Rule State** – toggles Decoupling Capacitor Distance to Via rules on or off.
- **Maximum Distance Between Capacitor and Via** – specifies the maximum distance between the capacitor pin and the via to the power or ground-reference plane.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.
- **Maximum Search Distance for Via** – specifies the maximum search distance for a via when a violation is found. This allows the user to quickly determine if the via is barely outside the original requirement and therefore not a serious violation.
- **Verify Via Connected to Appropriate Plane** – specifies whether EMI Scanner verifies the via connects to the selected plane (shape). Selecting 'no' speeds up processing.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Power/Ground Trace Decoupling** – displays the **Power/Ground-Reference Trace Decoupling** rule configuration.



This rule states that all power and ground-reference traces longer than a specified length that provide power to a fast switching chip must have a decoupling capacitor within a specified distance on the IC power pin.

From here, set the following:

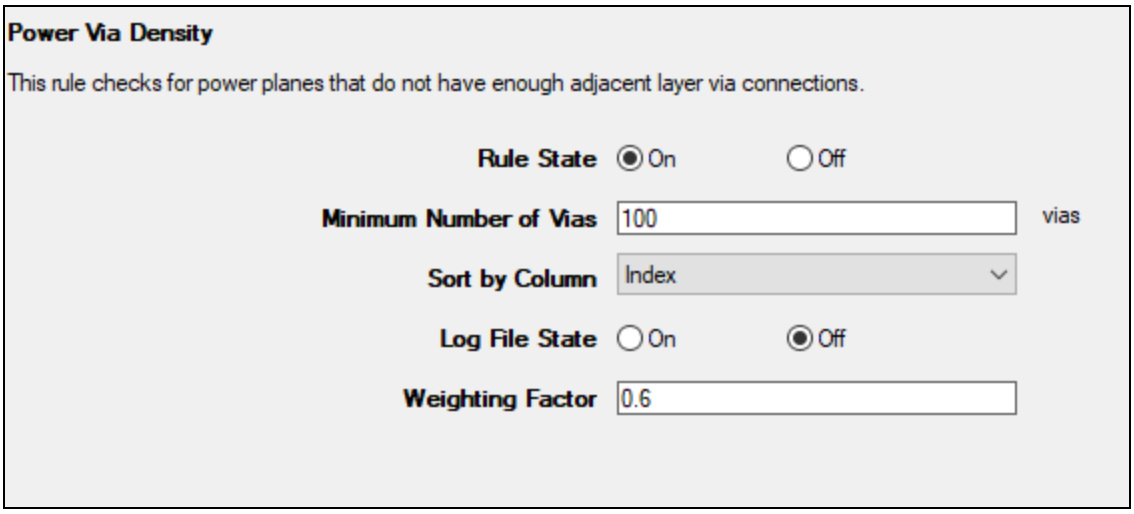
- **Rule State** – toggles Power/Ground-Reference Trace Decoupling rules on or off.
- **Maximum Ignorable Power/Ground Trace Length** – specifies the maximum allowable power/ground trace length that may be ignored. All traces longer than this length must meet this rule's requirement.
- **Max Distance Between Cap and IC Power Pin** – specifies the maximum distance between the capacitor pin and the power or ground-reference pin.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

- **Net Choice** – specifies the nets to include in this rule analysis. Select one, or **Ctrl**+click to select multiple nets.

- **Power Via Density** – displays the **Power Via Density** rule configuration.



This rule checks for power planes that do not have enough adjacent layer via connections.

From here, set the following:

- **Rule State** – toggles Power Via Density rules on or off.
- **Minimum Number of Vias** – sets the minimum number of vias required to connect adjacent power planes.
- **Sort by Column** – specifies the column by which results are sorted.
- **Log File State** – determines whether EMI Scanner creates a log file with additional details (intended for use in debugging).
- **Weighting Factor** – specifies the weighting factor (0<factor<=1) that is applied to violations of this rule when assigning a rank. A violation's rank is its own "degree of badness" multiplied by the weighting factor. See Exporting EMI Scanner Results and Analyzing EMI Scanner Results Using iQ-Harmony.

4. Rules are automatically saved for the current EMI scan. To save them for a future scan, click **Save Rules**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

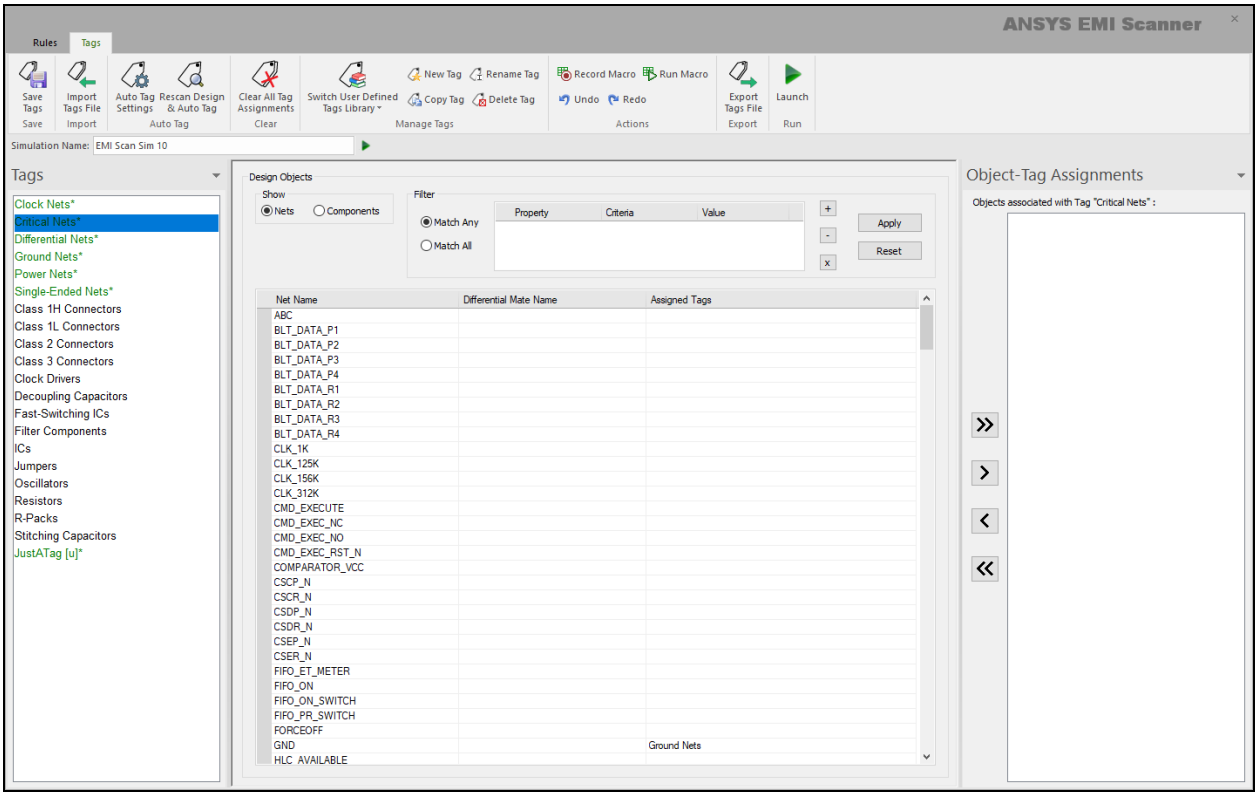
- You can display a description of a rule parameter at any time as with the cursor over it.
- Units of measure are set in General Settings.

Using Tags in EMI Scanner

The EMI Scanner **Tags** tab contains tag assignment panes and a ribbon with the following buttons:

- **Save Tags** – saves tags to the **Tags** pane.
- **Import Tags File** – allows you to import tags from a *.tgs file.
- **Auto Tag Settings** – allows you to change default tagging behavior.
- **Rescan Design & Auto Tag** – applies new tagging behavior.
- **Clear All Tag Assignments** – unassigns all nets and components from tags.
- **Switch User-Defined Tags Library** – allows you to browse to a different tagset.
- **New Tag** – creates a new tag.
- **Rename Tag** – renames a user-created tag.
- **Copy Tag** – copies a tag's properties into a new tag.
- **Delete Tag** – removes a tag on the **Tags** pane.
- **Record/Stop Macro** – starts or stops macro recording.
- **Run Macro** – runs a previously recorded macro.
- **Undo** – used to undo the previous action (only for manual assignments).
- **Redo** – used to restore an undone action (only for manual assignments).
- **Export Tags File** – allows you to export tags in a *.tgs file for later use.
- **Launch** – begins the EMI scan.

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 Tags pane (located on the left), Net tags appear in green, while Component tags appear in black. User-created tags are denoted by **[u]**.

Tags can be assigned automatically or manually.

Also add custom tags or create a tagging macro.

Assigning Tags Automatically

Upon loading, some tags are automatically assigned nets or components. By default, Auto Tag makes the following selections:

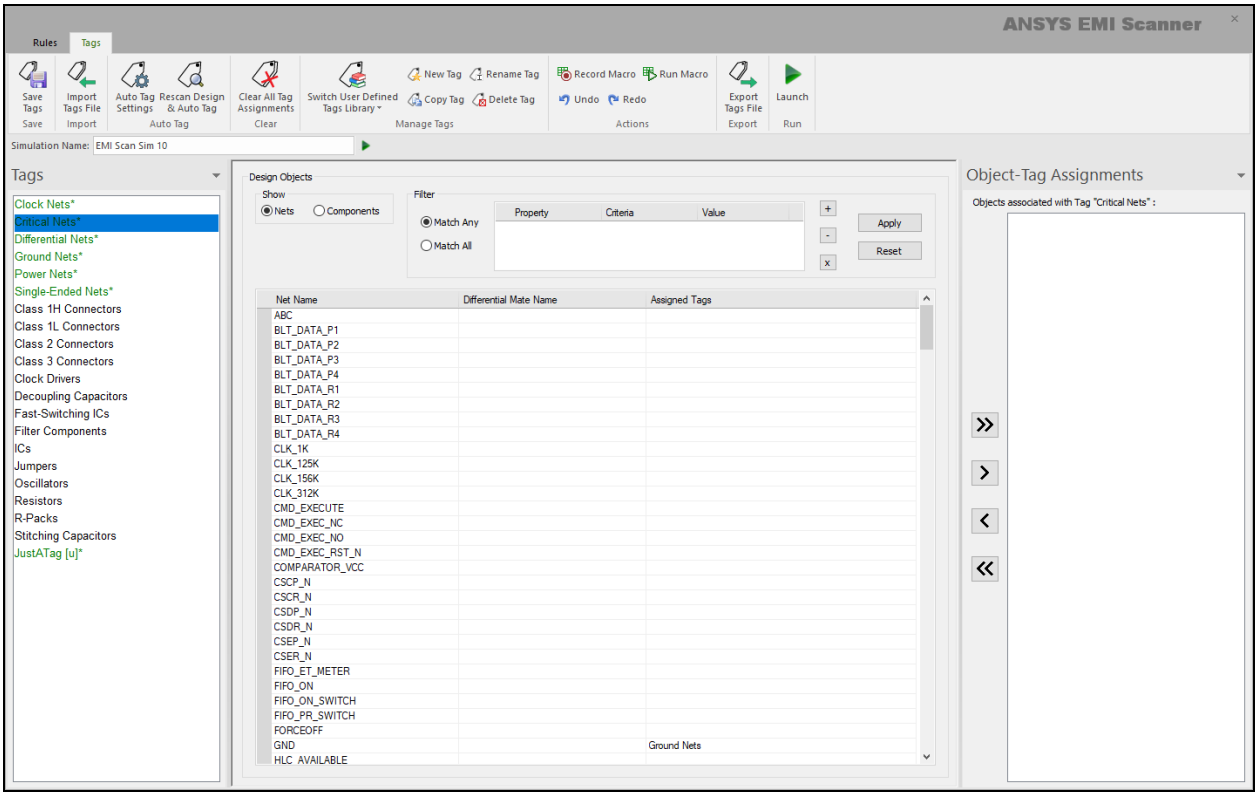
- **Critical Nets** – non power-ground nets that have a port.
- **Differential Nets** – those set up in the Differential Nets workspace.
- **Power/Ground Nets** – those set up in the Power/Ground workspace. Otherwise, distinguished by net name.
- **Decoupling Capacitors** – capacitors with names starting from C or EMC, with a value less than 10uF.
- **Stitching Capacitors** – capacitors connecting two power planes or two ground planes.

To change Auto Tag 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. Launch EMI Scanner.
2. Click **Tags**.

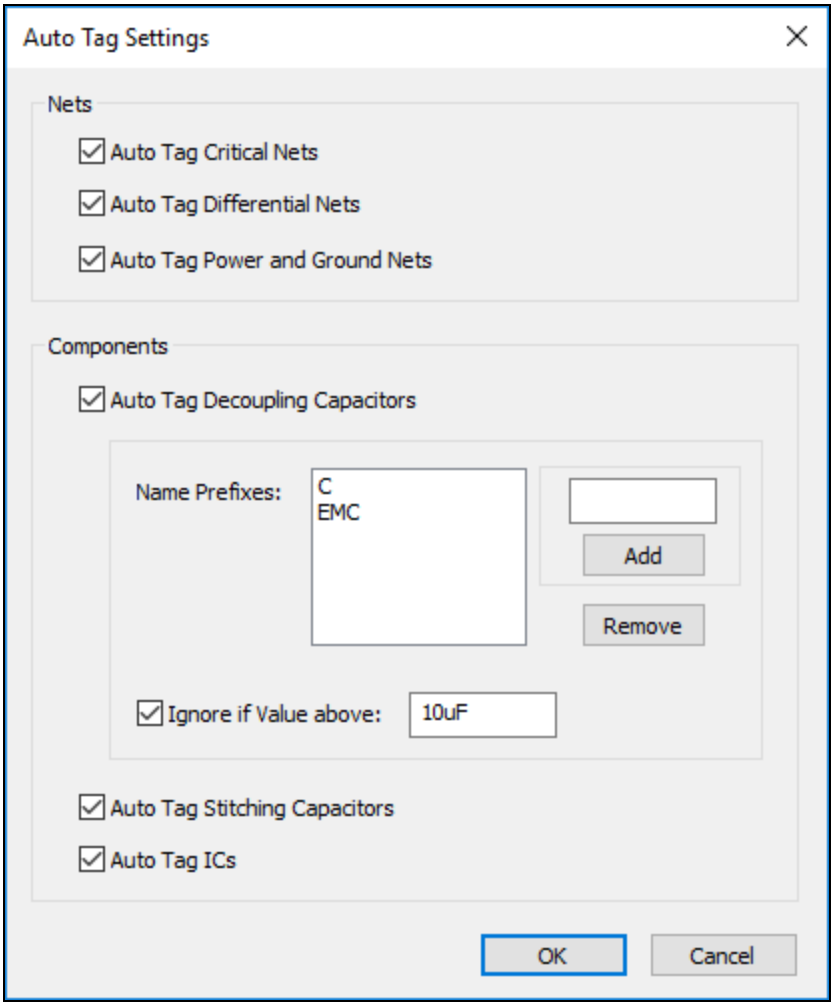
   The **EMI Scanner** displays the **Tags** tab.



   Net tags appear in green, while Component tags appear in black.

3. From the **Auto Tag** area, click **Auto Tag Settings** to open the **Auto Tag Settings** 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. From the **Nets** area, determine the types of nets you want automatically tagged.
5. From the **Components** area, use the check boxes to determine whether to automatically tag Decoupling Capacitors, Stitching Capacitors, or ICs.

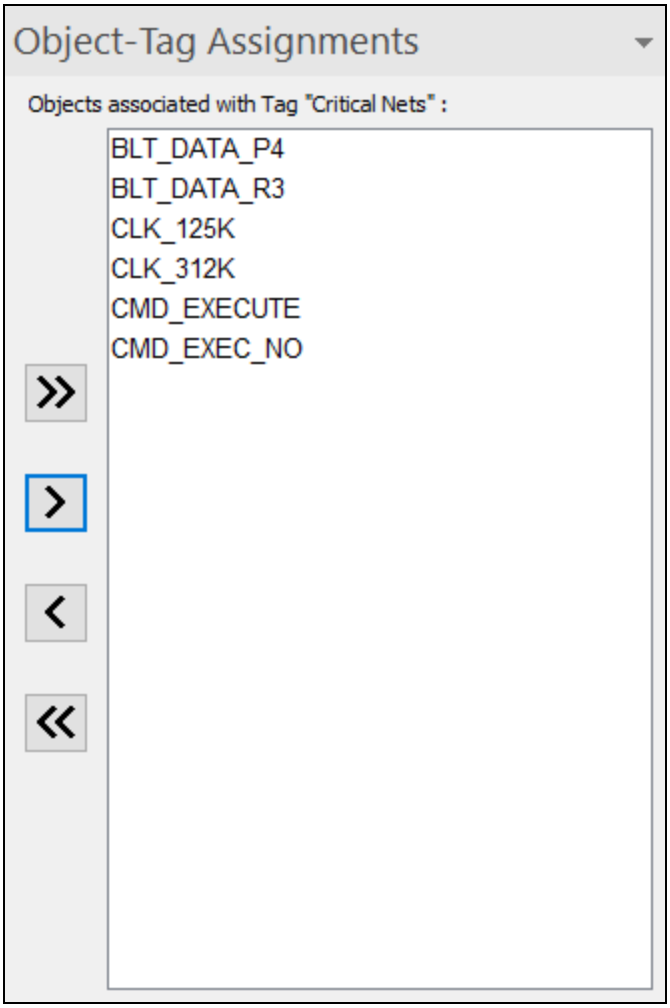
   If you select **Auto Tag Decoupling Capacitors**, select to **Add** or **Remove** name prefixes using the list and field. Also select **Ignore if Value above** to ignore capacitors above the value you specify.

6. Click **OK** to apply settings and return to the **EMI Scanner** window.
7. Click **Rescan Design & Auto Tag**.

   The design is scanned and tagged according to your settings.

   The **EMI Scanner** window displays the new tags in the **Object-Tag Assignments** pane for each tag.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



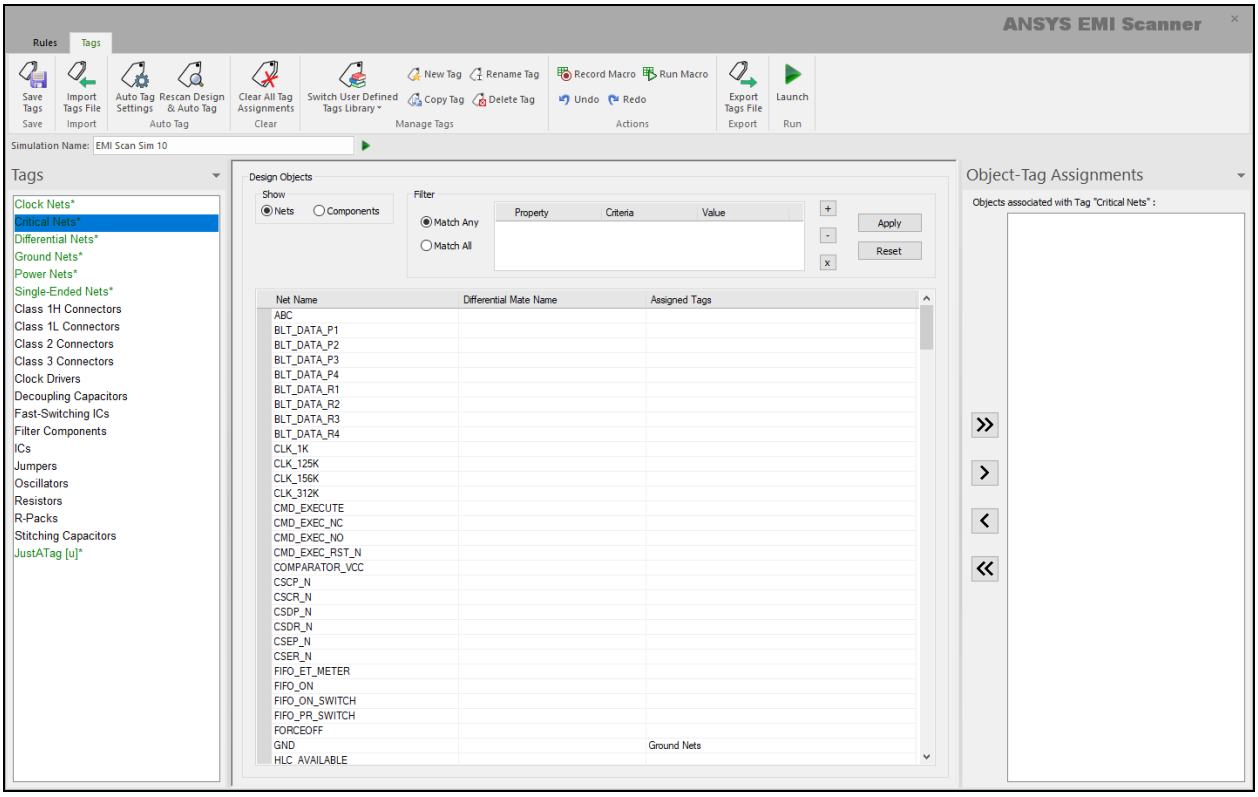
Assigning Tags Manually

To assign tags manually:

1. Launch EMI Scanner.
2. Click **Tags**.

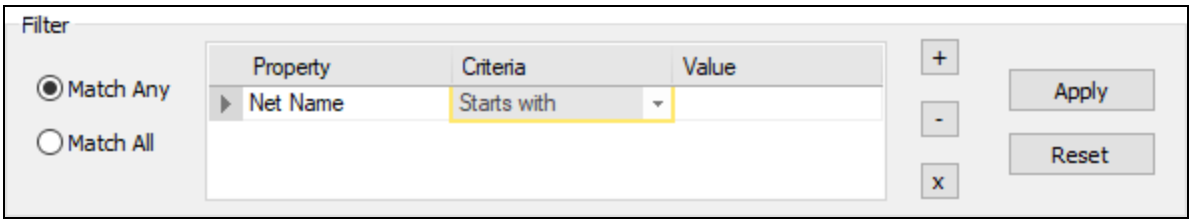
   The **EMI Scanner** displays the **Tags** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Net tags appear in green, while Component tags appear in black.

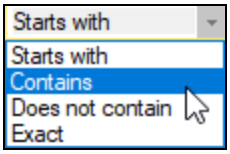
3. From the **Tags** pane, select a tag (e.g., Clock Nets, Decoupling Capacitors, Resistors) to which you want to assign nets or components. Also add custom tags.
4. The center pane updates to display a list of eligible nets or components. If selected, apply filters to view only the nets or components you're looking for:



   a. Select either **Match Any** (nets or components matching any one criterion displays) or **Match All** (only nets or components meeting all criteria displays).
   b. Add or remove rows on the filter list by clicking **+** and **-**. Also clear all criteria by clicking (**X**).

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 **Criteria** drop-down menu to select the filter type.



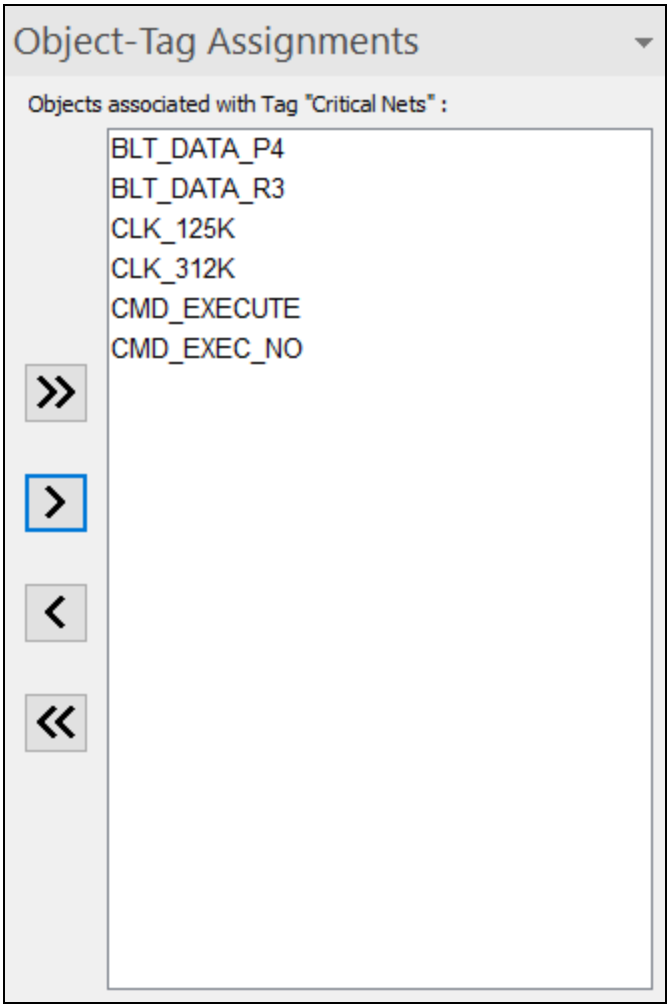
d. Enter filter values in the **Value** field.

e. Click **Apply** to apply the filters and update the net/component list.

f. Click **Reset** to remove the filter and display the default nets or components.

5. Select nets or components in the list, and use the right arrow button (**>**) to assign them. Use the double arrow (**>>**) to assign all listed nets or components to that tag.

   Assigned nets or components appear in the **Object-Tag Assignments** pane.



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 unassign nets or components individually by selecting them and using the left arrow (**<**), or unassign all nets or components using the double arrow (**<<**).

6. If selected, click **Export Tags File** to save your tags as a *.tgs file. You can later click **Import Tags File** to use these settings again.

Working with Custom Tags

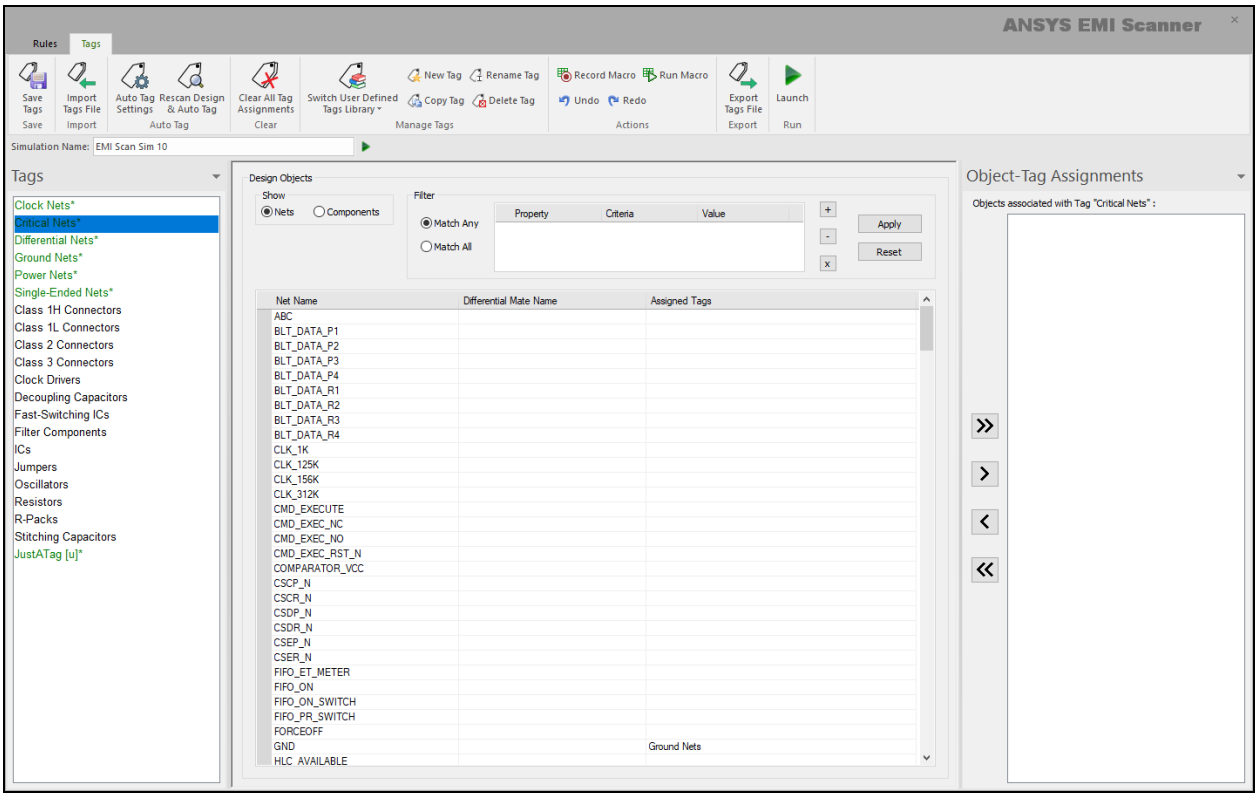
EMI Scanner also allows you to create and assign custom tags. User-created tags are denoted by **[u]** in the Tags pane, and can be assigned manually.

Adding Custom Tags

To add tags:

1. Launch EMI Scanner.
2. Click **Tags**.

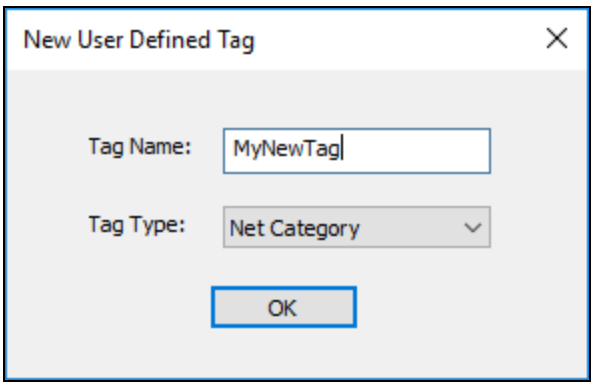
   The **EMI Scanner** displays the **Tags** tab.



Tags are listed in the Tags pane on the left. Net tags appear in green, while Component tags appear in black.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

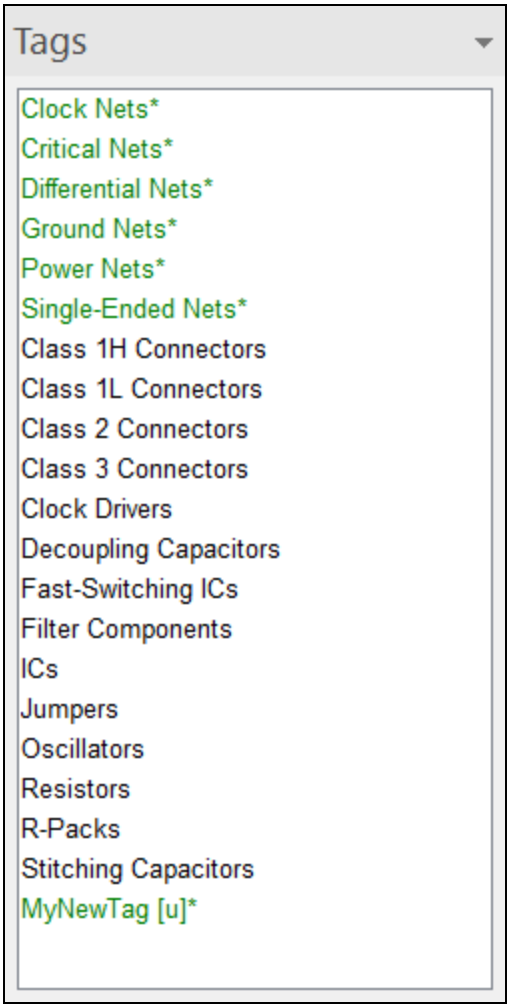
3. From the **Manage Tags** area of the ribbon, select **New Tag** to open the **New User-Defined Tag** window.



4. Enter a **Tag Name** and select the **Tag Type**. You can create a tag either in the **Net Category** or **Component Category**.
5. Click **OK** to return to the **EMI Scanner** window.

   The new tag appears at the bottom of the **Tags** pane, with a **[u]** indicating that it is a user-created tag.

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 save your tags, click **Save Tags**.

   Your tags appears in the **Tags** pane the next time you launch EMI Scanner.

Also create a custom tag from an existing tag. To do so:

1. Select a tag on the **Tags** pane.
2. From the **Manage Tags** area, click **Copy Tag**.

   The **Copy User-Defined Tag** window opens, with the **Tag Type** already selected.

3. Name the new tag and click **OK**.

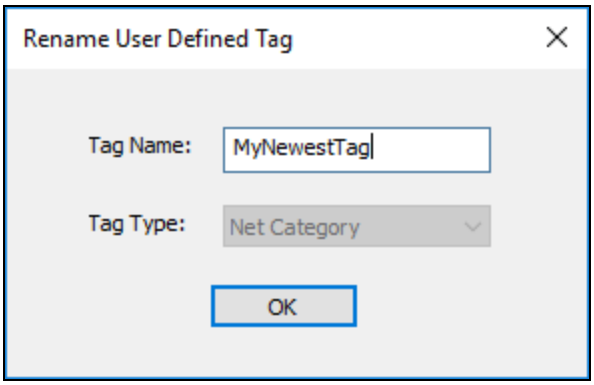
   The new tag appears in the **Tags** pane.

Renaming a Custom Tag

To rename a custom tag:

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 tag on the **Tags** pane.
2. From the **Manage Tags** area, select **Rename Tag** to open the **Rename User-Defined Tag** window.



3. Rename the tag and click **OK**.

> **Note:**
>
> You can only rename custom tags. You cannot rename a default tag.

Deleting a Custom Tag

To delete a custom tag:

1. Select the tag on the **Tags** pane.
2. Use the double left arrow (**<<**) in the **Object-Tag Assignments** pane to remove any assignments.

> **Important:**
>
> You cannot delete a tag until all object assignments are removed from it.

3. From the **Manage Tags** area, click **Delete Tag**.

   The tag disappears on the **Tags** pane.

> **Note:**
>
> You can only delete custom tags. You cannot delete a default tag.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

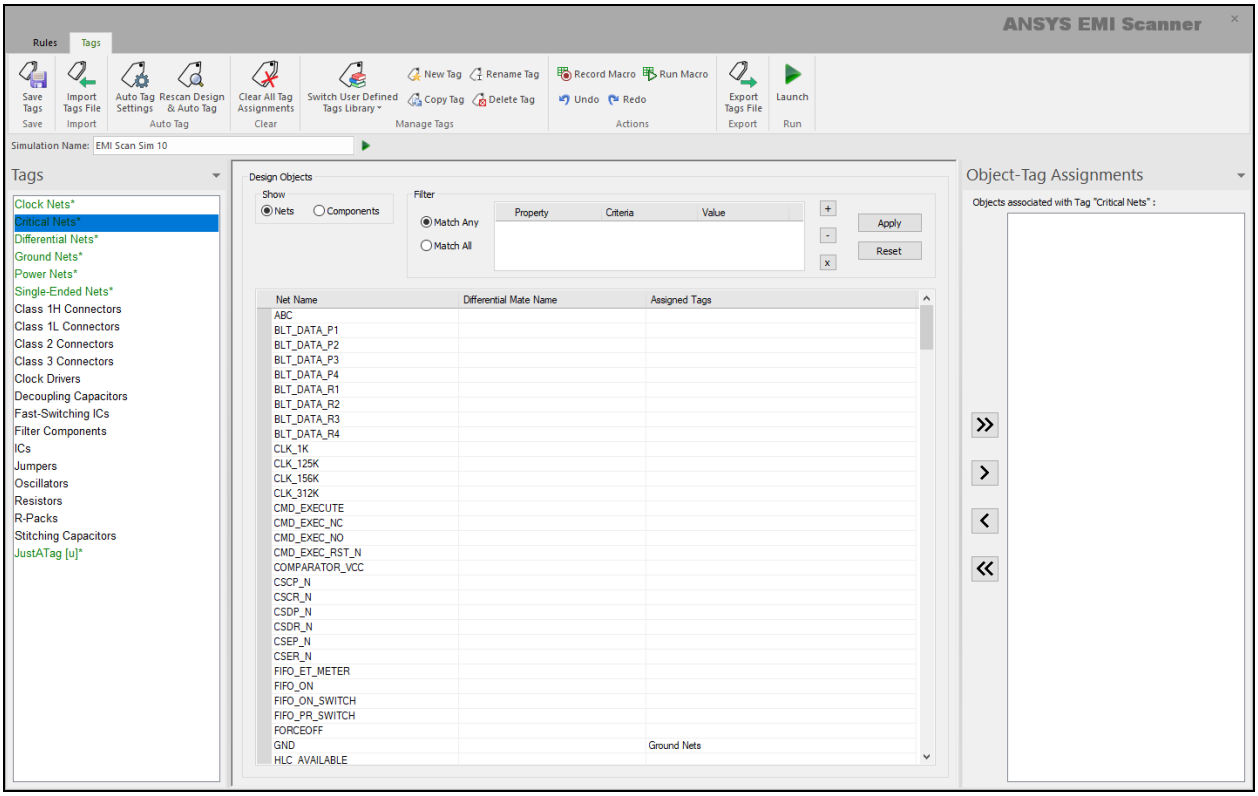
Creating Tagging Macros

You can save your tagging process as a series of steps. Macros are saved as XML-based *.tmx files.

To create a macro:

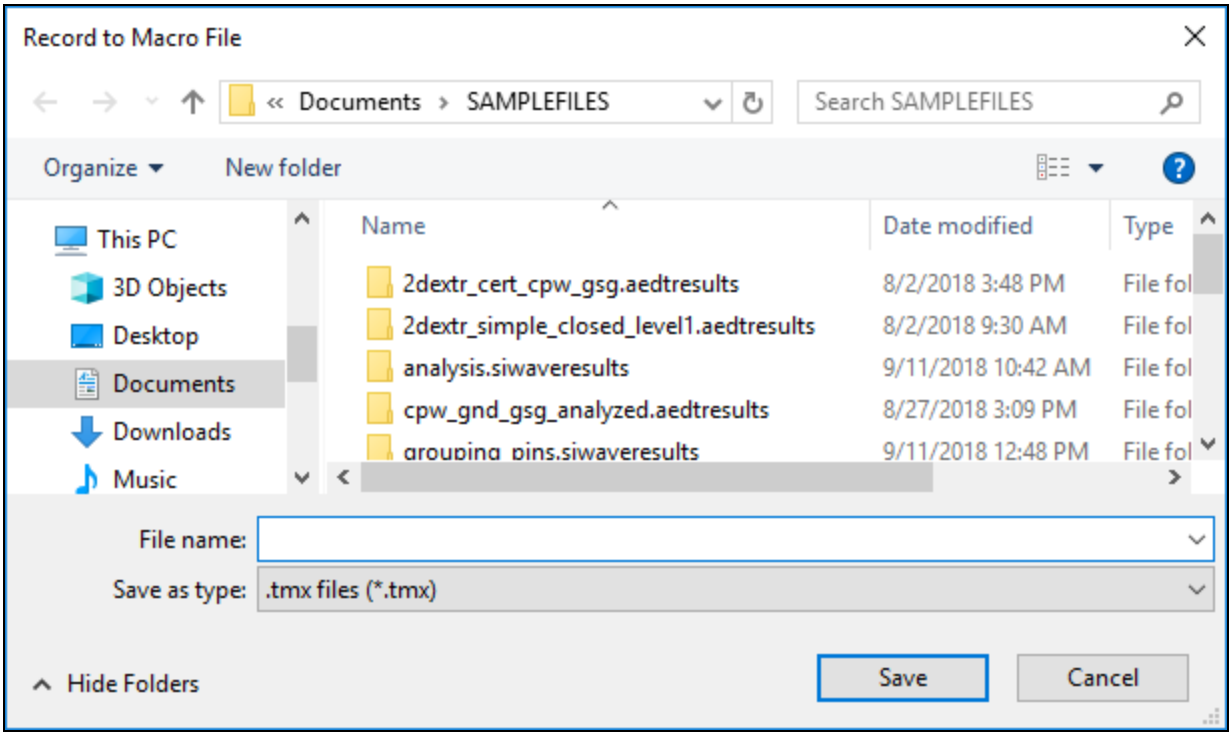
1. Launch EMI Scanner.
2. Click **Tags**.

   The **EMI Scanner** displays the **Tags** tab.



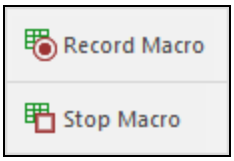
3. From the **Actions** area of the ribbon, select **Record Macro** to open the **Record to Macro File** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Navigate to your selected save location, enter a **File name**, and click **Save**.

   Record Macro becomes **Stop Macro**.



5. Complete the appropriate actions to save as a macro. You can **Undo** or **Redo** actions using the buttons in the Actions area.
6. When you are finished, click **Stop Macro**.

   The macro *.tmx file is saved at the location you specified earlier.

To run a saved macro, click **Run Macro**.

> **Note:**
>
> You cannot record and run a macro at the same time.

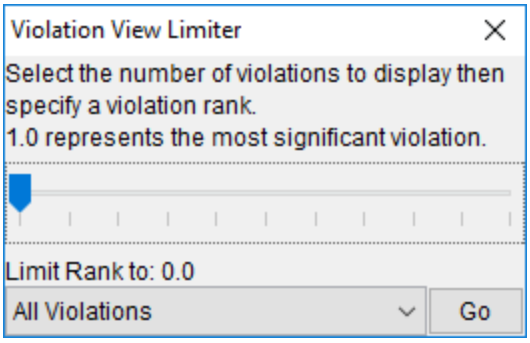
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analyzing EMI Scanner Results Using iQ-Harmony

iQ-Harmony receives violations on the EMI Scanner and, based on signal characteristics, makes predictions on the seriousness of the violations. This offers meaningful comparison between rule limits and violations, in order to make "what if" comparisons and determine impacts.

To analyze EMI Scanner results:

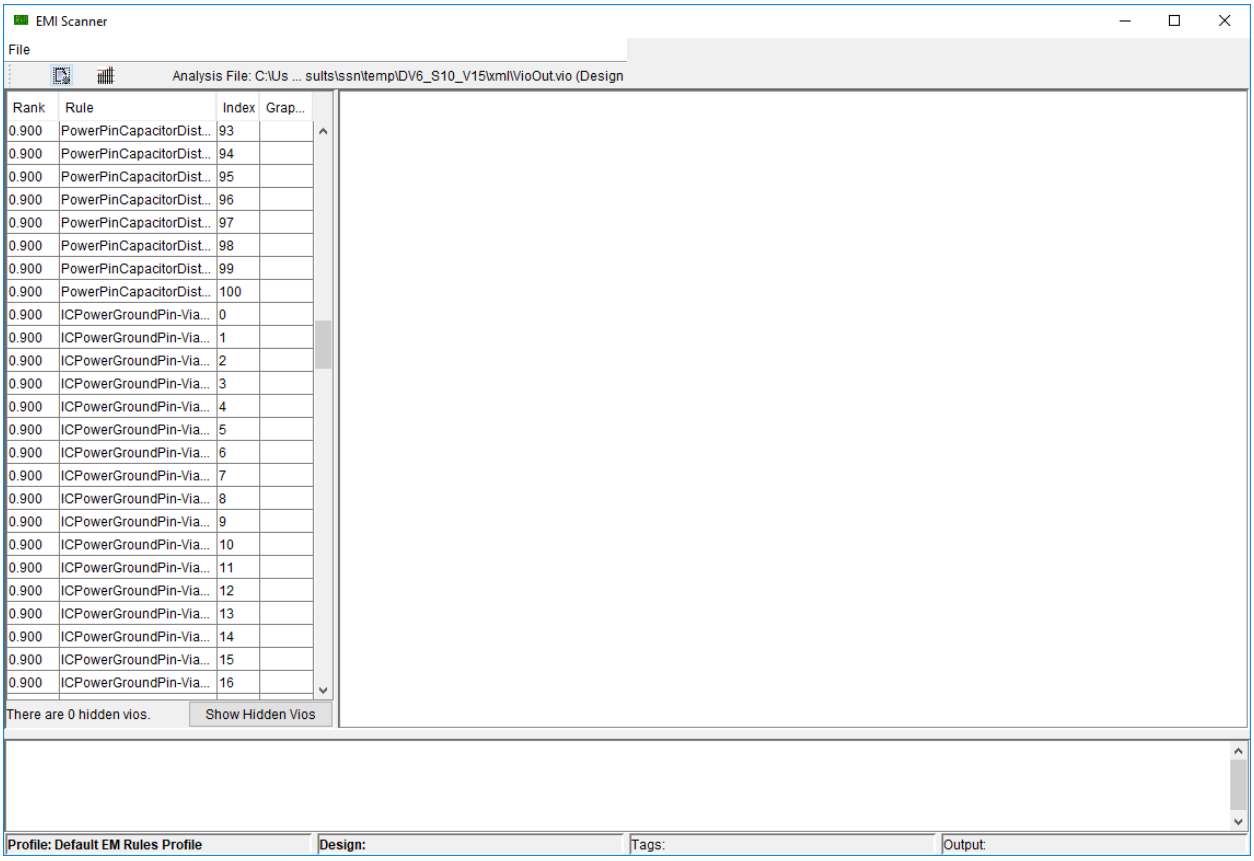
1. Run an EMI scan.
2. From the **Project Manager** window, right-click **EMI Setup** and select **Results** > **Analyze with iQ-Harmony** to open the **Violation View Limiter** window.



3. Use the drop-down menu to select the number of violations to display. The default is **All Violations**.
4. Use the slider to select a minimum violation rank to display. 0.0 is the least severe and 1.0 is the most severe.

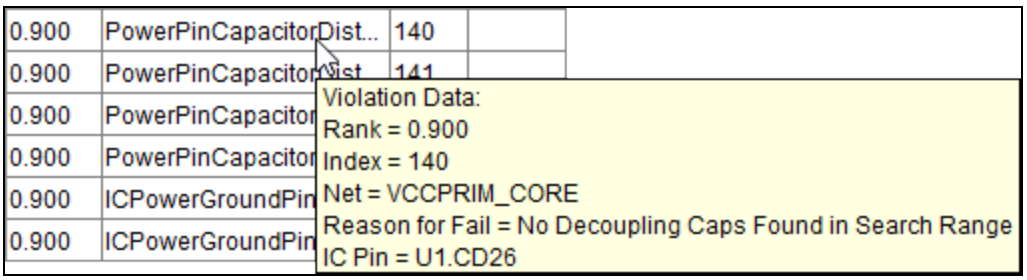
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Go** to open the **EMI Scanner** window.



Violations are listed in the left-hand pane. By default, they are sorted by rank, with the most severe violations at the top of the list. Click the column headings to change the sorting.

Hovering the cursor over a violation offers more information about the violation.

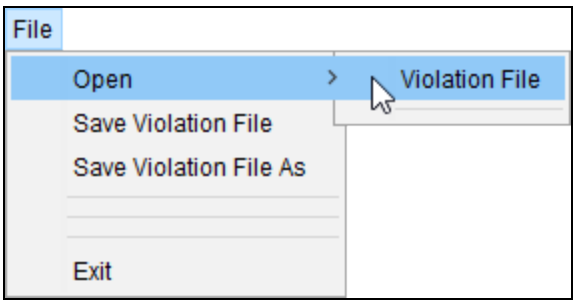


If there are any hidden violations, text at the bottom of the pane notes this, and offers you the option to **Show Hidden Vios**.

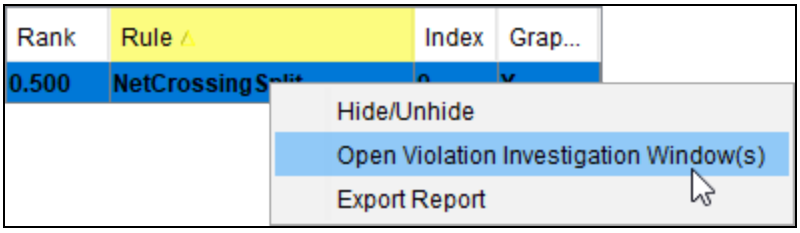
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **File** menu allows you to save the violations list as a *.vio file, or open an existing *.vio file.

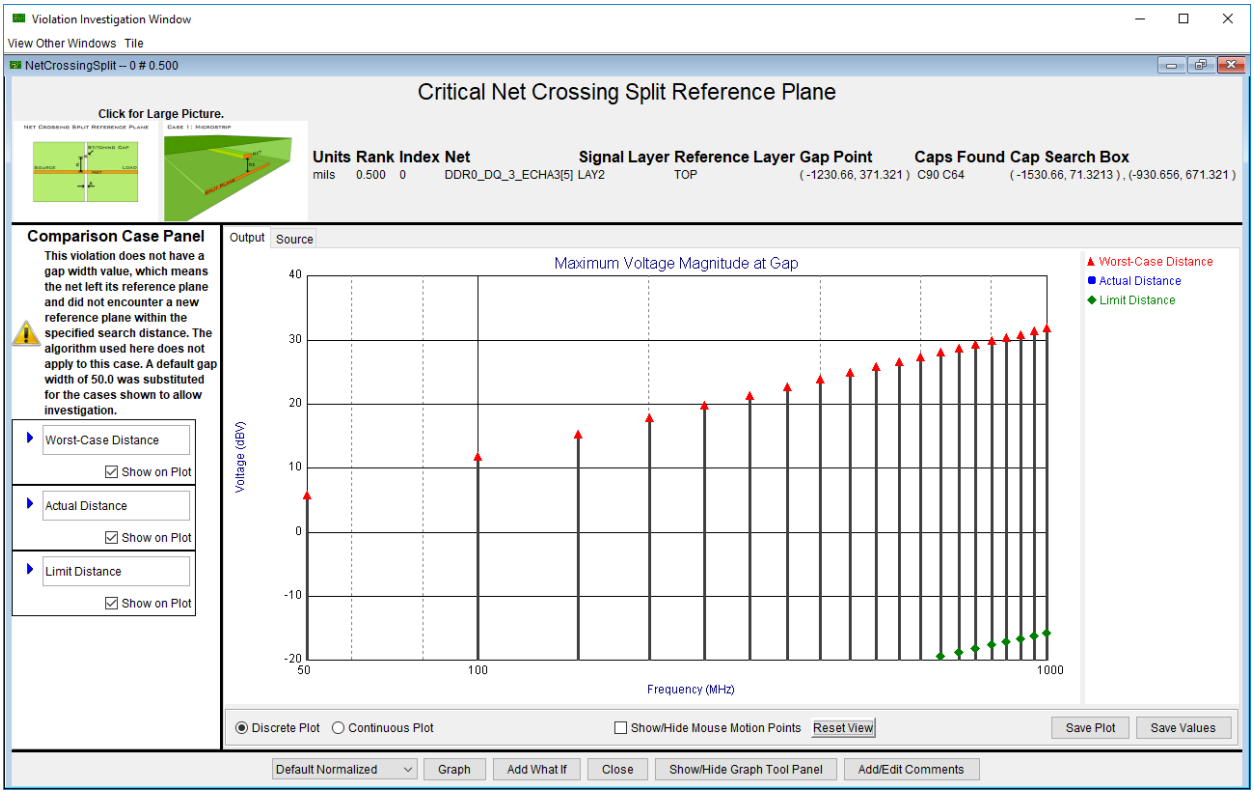
Also open an existing *.vio file by clicking the **Open Analysis File** ( ) icon.



The **Current Spectra Library** icon ( ) opens the **Spectra Configuration** window. From the left pane, either double-click a violation, or right-click it and select **Open Violation Investigation Window(s)** to open the **Violation Investigation** window.



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 of the **Violation Investigation** window displays information about the violation's location and severity, and graphic representations of the type of violation. Click a graphic to enlarge it for easier viewing.

The left pane is the **Comparison Case Panel**. The default cases vary depending on the type of violation, and can include: Worst-Case Distance, Actual Distance, Limit Distance, Search Limit Distance. Click the blue triangles to expand the cases and view their display options. Add a comparison case by clicking **Add What If**.

The comparison cases allow you to see how changes in signal characteristics affects the violations. Use the **Show on Plot** check boxes to determine which cases are displayed. You can rename a case by typing into its name field.

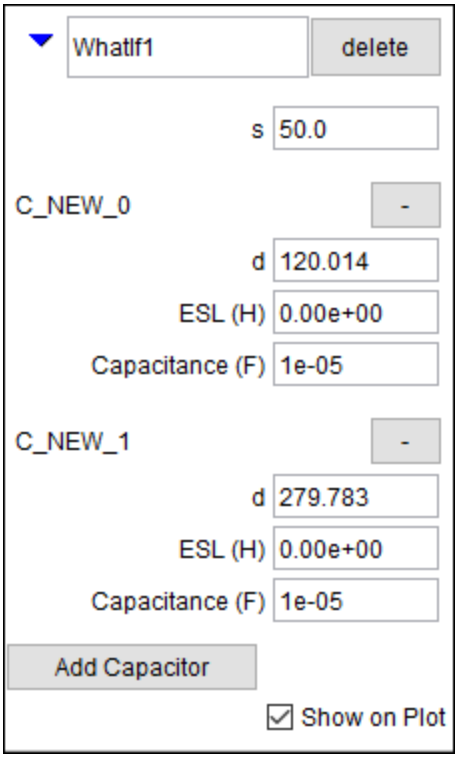
The bottom of the **Violation Investigation** window allows you to make additional changes.

Options include:

- **Choose A Spectrum** – allows you to select a default spectrum or one you have added via Spectra Configuration.
- **Graph** – after you change data, click this button to refresh the graph.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Add What If** – adds a new comparison case to the Comparison Case Panel.

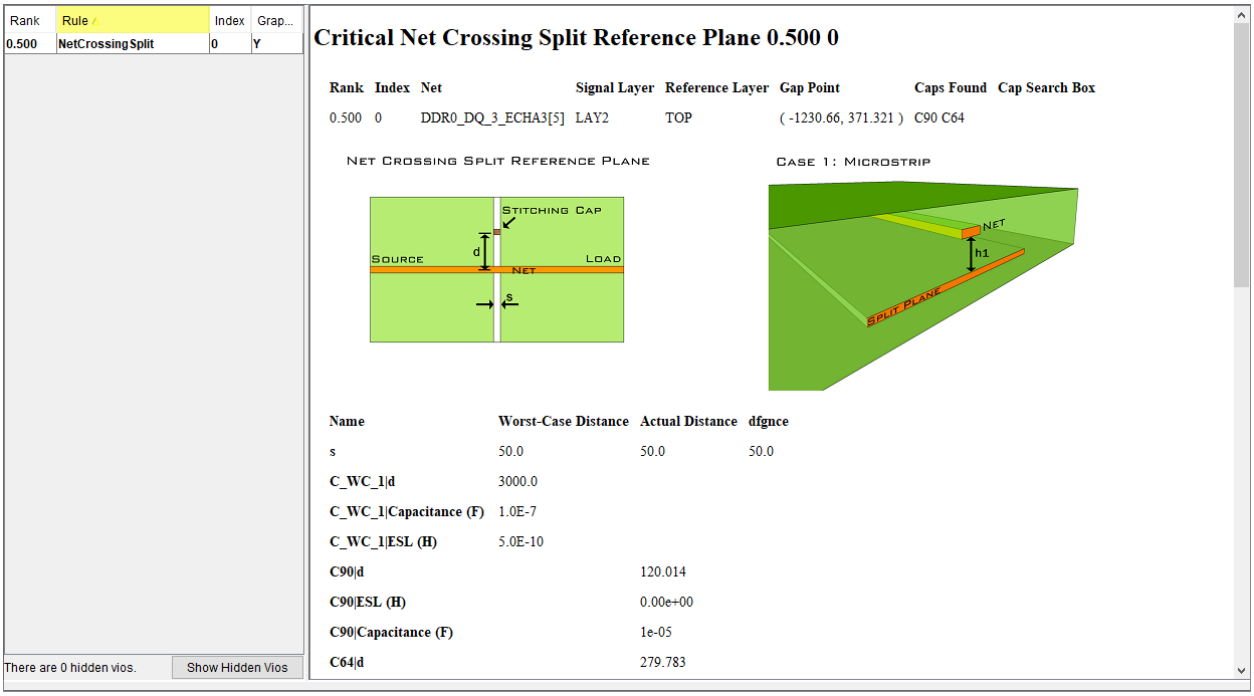


Change the settings and rename the case just as with default comparison cases. Also delete the custom case.

- **Close** – closes the Violation Investigation Window and returns you to the EMI Scanner window.
- **Show/Hide Graph Tool Panel** – toggles the display of the Graph Tool Panel:
- **Add/Edit Comments** – opens a blank text window to type comments. These are saved with the *.vio file.

6. Analyze the data as selected.
7. Click **Save Plot** to save the plot as a *.jpeg or *.png image.
8. Click **Save Values** to save the values in plain text format.
9. Click **Close** to close the **Violation Investigation** window closes, and the **EMI Scanner** window displays your analysis in the right pane.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

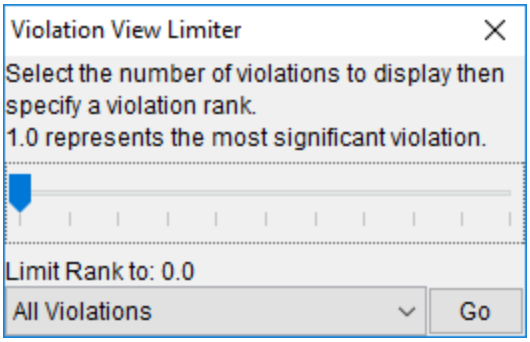


10. When you are finished analyzing, click **File** > **Save Violation File** or **File** > **Save Violation File As** to save your analysis in *.vio format.

Spectra Configuration

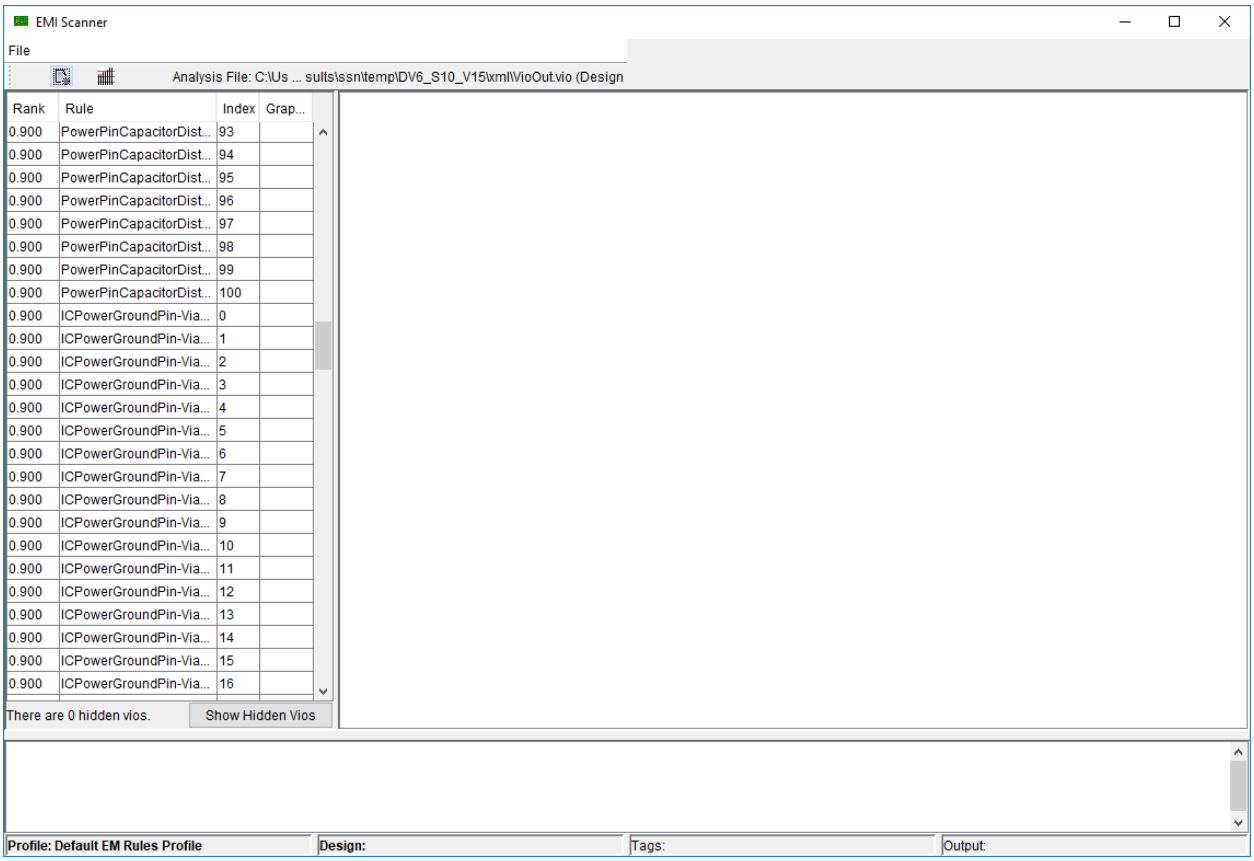
To configure the Spectra library:

1. Run an EMI scan.
2. Click **Results**.
3. Use the EMI Scan drop-down menu to select **[Simulation Name]** > **Analyze with iQ-Harmony** to open the **Violation View Limiter** window.



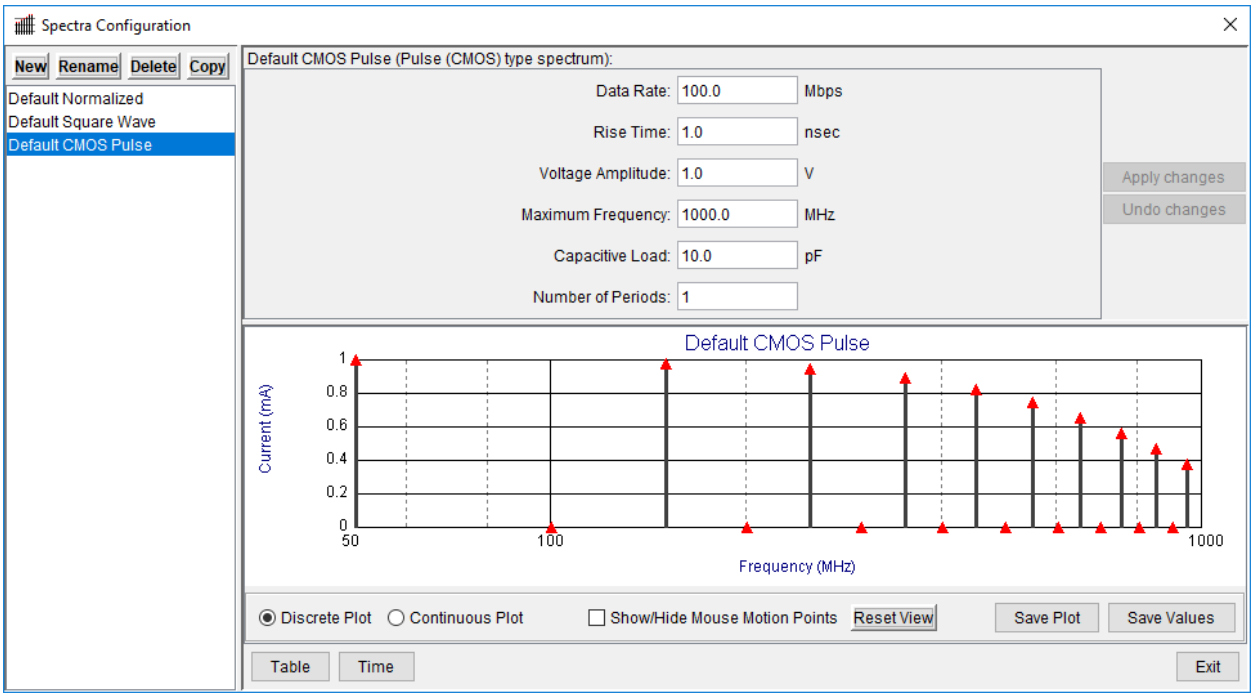
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Use the drop-down menu to select the number of violations to display. The default is **All Violations**.
5. Use the slider to select a minimum violation rank to display. 0.0 is the least severe and 1.0 is the most severe.
6. Click **Go** to open the **EMI Scanner** window.



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 the **Current Spectra Library** ( ) icon to open the **Spectra Configuration** window.



Select a spectrum in the left pane to view its settings and plot to the right. You can edit fields directly, then **Apply changes** or **Undo changes**.

By default, a Frequency/Current plot appears. The plot can be viewed as a **Discrete Plot** or a **Continuous Plot**.

You can **Save Plot** as a *.png or *.jpeg image, or **Save Values** in plain text format.

Click **Table** to view information as a table instead of a graph.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

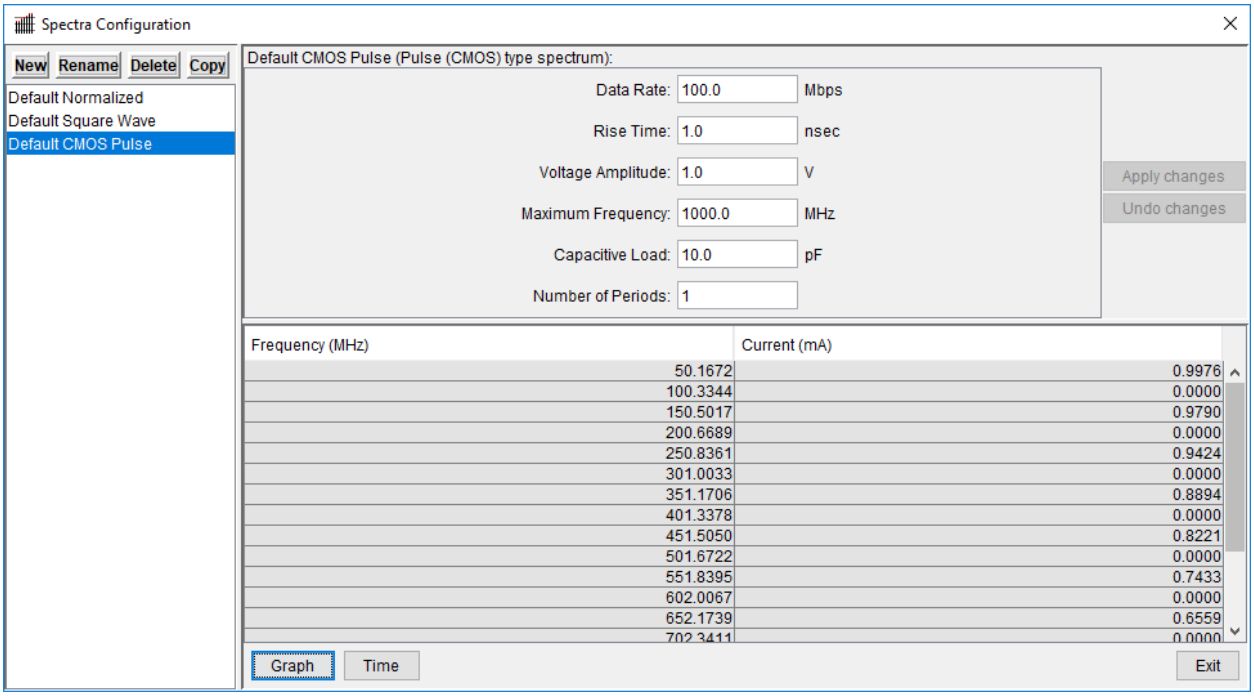
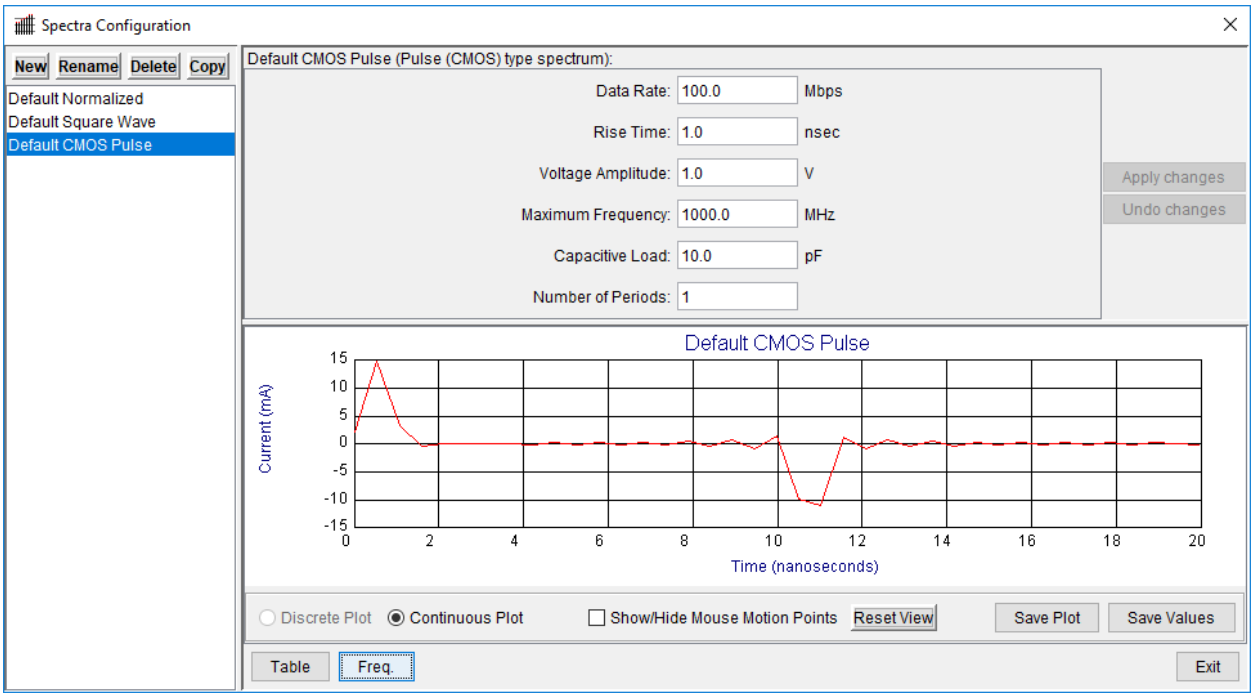


Table becomes **Graph**. Click it again to restore the plot view.

Click **Time** to view Current/Time information instead of Current/Frequency.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Time becomes **Freq.**. Click it again to return to Current/Frequency.

From the Spectra Configuration window, you can also perform the following actions:

- **New** – create a new spectrum.
- **Rename** – change a spectrum name.
- **Delete** – remove a spectrum.
- **Copy** – create a copy of an existing spectrum with a new name.

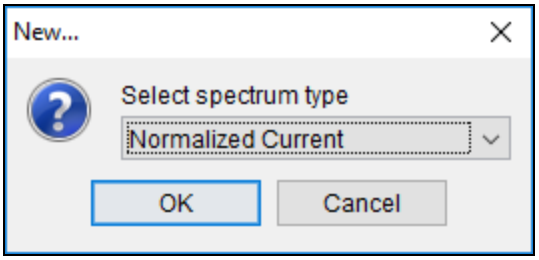
8. Click **Exit** to return to the **EMI Scanner** window.

   Any changes you made are now available in the **Violation Investigation** window.

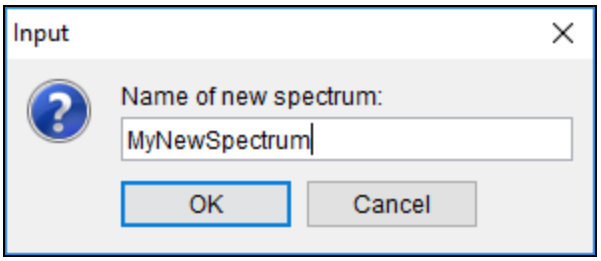
Creating a New Spectrum

To create a new spectrum on the Spectra Configuration window:

1. Click **New** to open the **New** window.



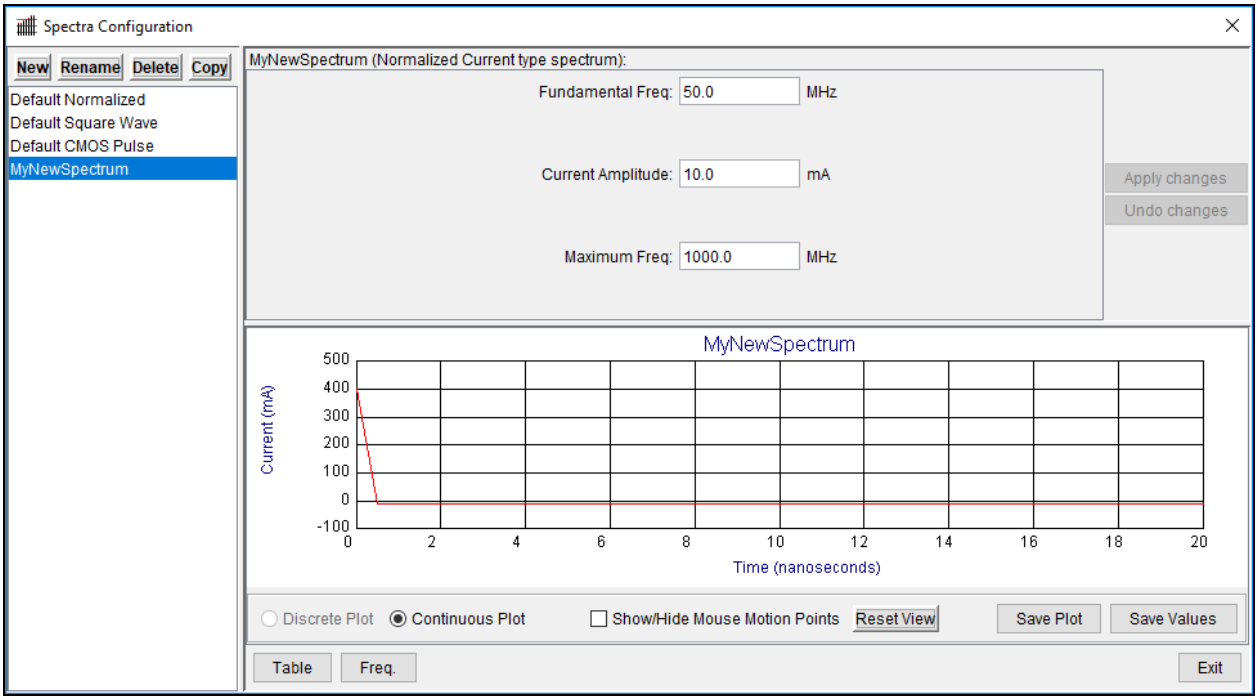
2. Use the drop-down menu to select an existing spectrum type (**Normalized Current**, **Square Wave**, or **Pulse CMOS**), or select **User-defined** to import settings.
3. Click **OK** to open the **Input** window.



4. Name the new spectrum and click **OK**.

   The new spectrum appears in the left pane, and its settings and plot appear to the right.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Configurable settings vary based on the type of spectrum you selected previously.

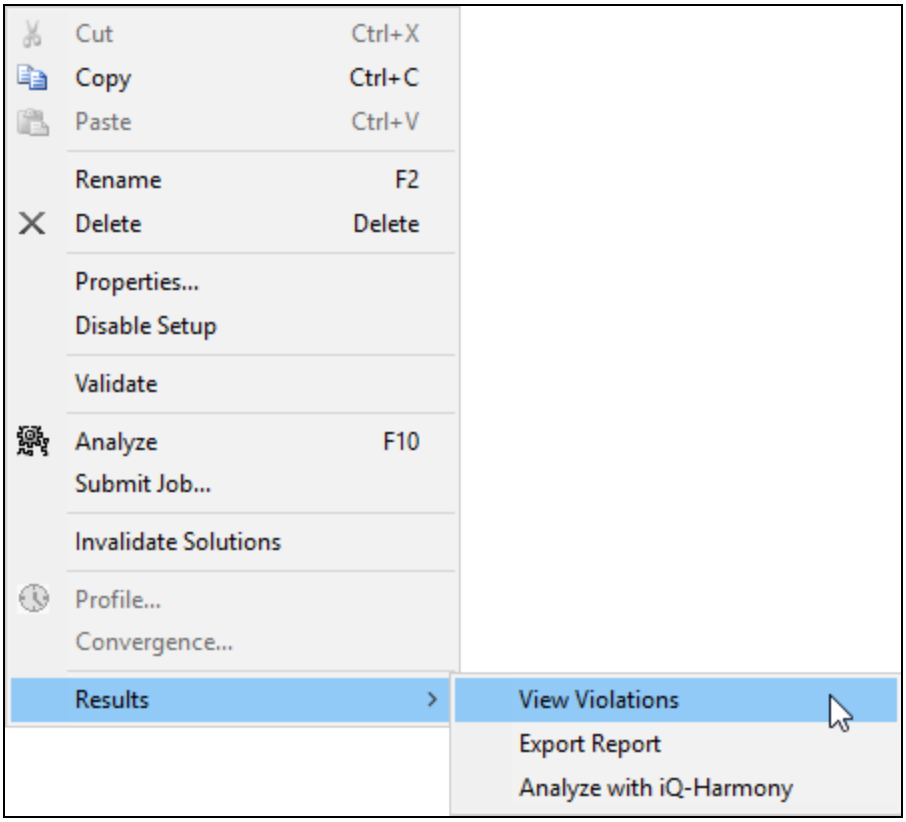
5. Enter the applicable settings and click **Apply Changes**.
6. Click **Exit** to return to the **EMI Scanner** window.

Viewing EMI Scanner Results

Follow these steps to view EMI Scanner results.

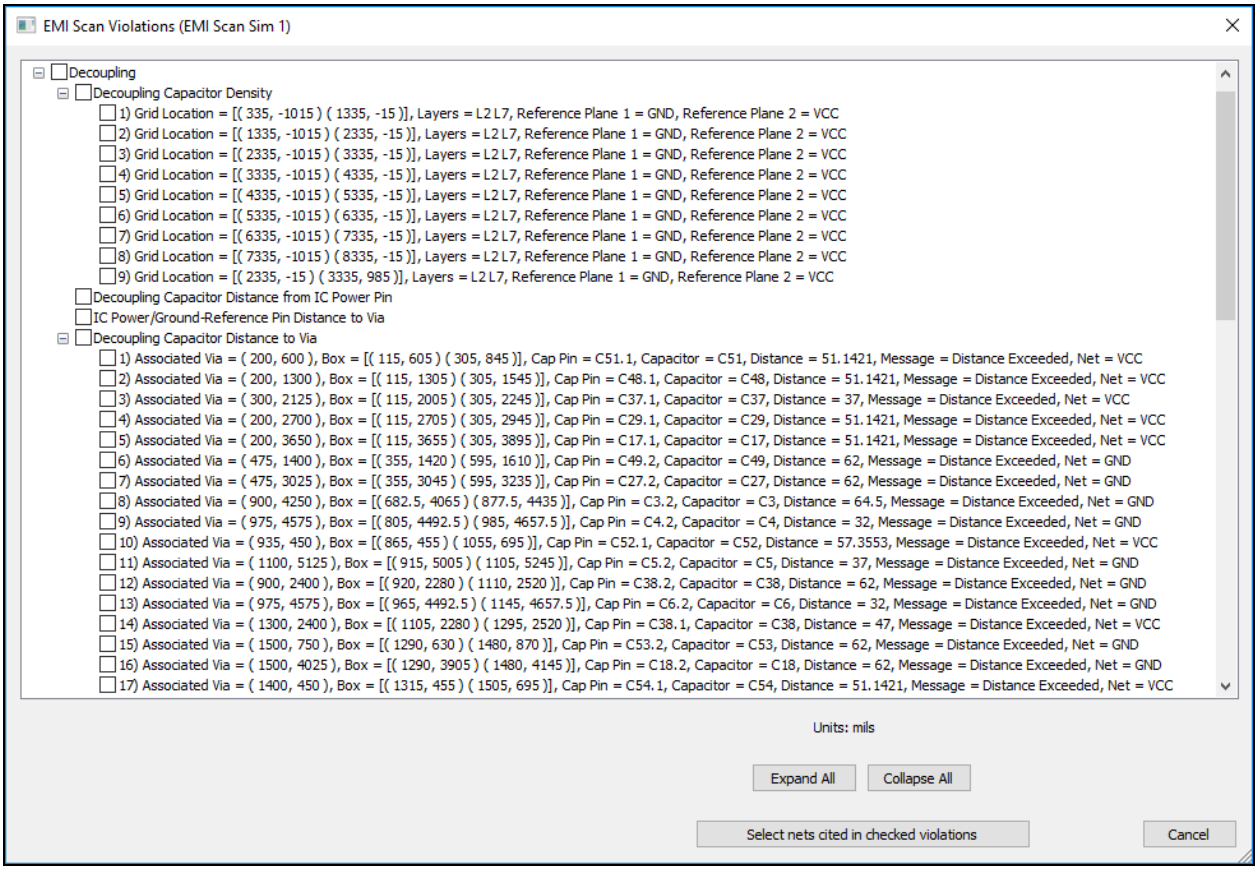
1. From the **Project Manager** window, right-click **EMI Setup** and select **Results** > **View Violations**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



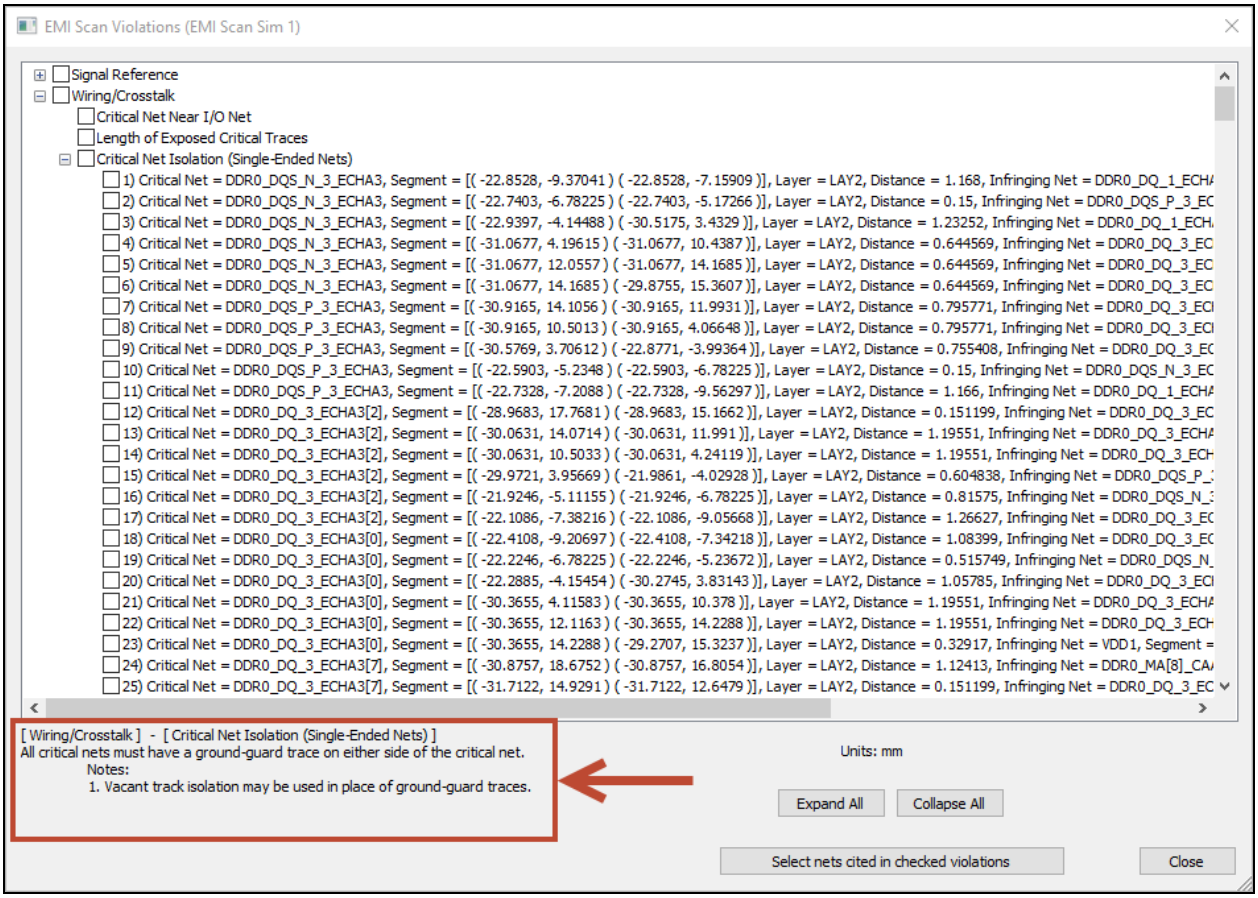
The **EMI Scan Violations** window opens.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Violations are listed in a tree, by type. Expand and collapse items in the tree, or click **Expand All**/**Collapse All**.
3. Click a violation to zoom to its location in the **Design Area**. Clicking a violation also populates the **EMI Scan Violations** window with a description of the violation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Use the check boxes in the tree to select multiple violations.
5. Click **Select nets cited in checked violations** to select these nets in the *Modeling* workspace.

Other Results Menu options include:

- **Export Report** – allows you to export an HTML report of EMI rules and violations.
- **Analyze with iQ-Harmony** – opens the Violation View Limiter window, the first step in analyzing EMI Scanner results.

Exporting EMI Scanner Results

To export EMI Scanner results:

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** window, right-click **EMI Setup**.

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 **Results** > **Export Report** to open an HTML report in the default Internet browser.

ANSYS EMI Scanner RULES

ANSYS EMI Scanner RULE RESULTS

SIGNAL REFERENCE

Critical Net Crossing Split Reference Plane - Violations: 0	ON
Critical Net Changing Reference Plane - Violations: A - 6, B - 0, C - 0	ON
Critical Net Near Edge of Reference Plane - Violations: 0	ON

WIRING/CROSSTALK

Critical Net Near I/O Net - Violations: 0	ON
Length of Exposed Critical Traces - Violations: 10	ON
Critical Net Isolation (Single-Ended Nets) - Violations: 0	ON
Critical Net Isolation (Differential Nets) - Violations: 0	ON
Critical Differential Net Length Matching and Spacing - Violations: 0	ON
Wide Power/Ground Traces - Violations: 434	ON

DECOUPLING

Decoupling Capacitor Density - Violations: 0	ON
Decoupling Capacitor Distance from IC Power Pin - Violations: 101	ON
IC Power/Ground-Reference Pin Distance to Via - Violations: 178	ON
Decoupling Capacitor Distance to Via - Violations: 146	ON
Power/Ground-Reference Trace Decoupling - Violations: 0	ON
Power Via Density - Violations: 1	ON

PLACEMENT

I/O Filter Distance to I/O Connector - Violations: 0	ON
Distance from Oscillator to Clock Driver - Violations: 0	ON

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 change whether EMI Scanner produces one large, combined HTML report or several individual HTML files in the EMI Scanner General Settings.

Nexxim Solutions in HFSS 3D Layout (General Mode Only)

Nexxim is a simulator for high-speed, high-density circuit designs, some of which can be analyzed through HFSS 3D Layout.

HFSS 3D Layout contains the following Nexxim solution types:

- LNA Solver
- Transient Analysis Solver
- AMI Analysis
- QuickEye Solver
- VerifEye Solver

Add LNA Solution Setup

A Nexxim Linear network analysis combines the separate analyses in an ECAD-MCAD cosimulation. Each component in the assembly is solved independently using its designated field solver (HFSS, Q3D, or SIwave) and the LNA then ties all the separate subcircuit models together to compute the end-to-end response of the assembled system. Time Domain Reflectometer (TDR) plots are available for LNA and imported solutions in HFSS 3D Layout.

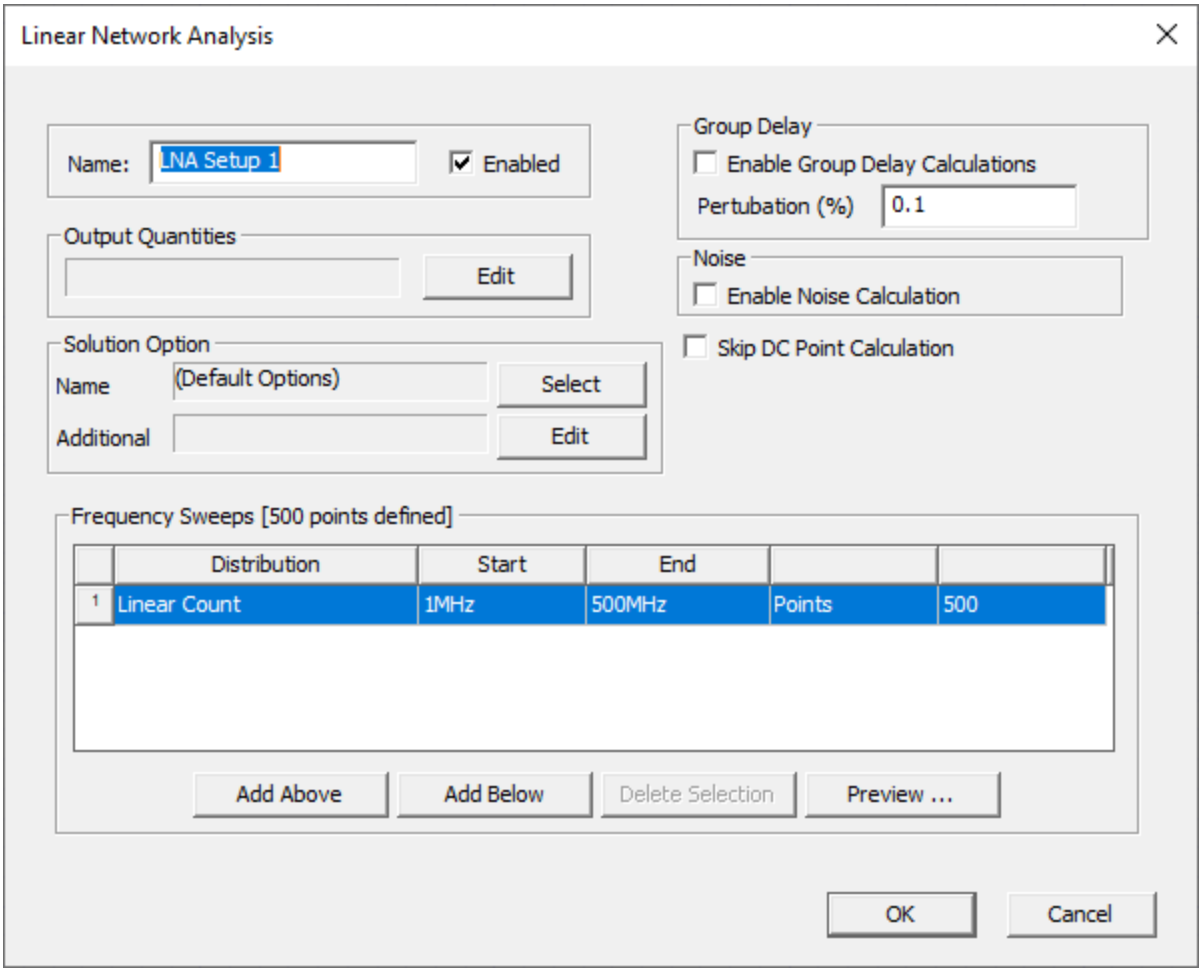
Nexxim Linear network analysis performs a frequency-domain analysis of the circuit that is linearized around the DC operating point. In addition to the basic linear analysis, you can also run a DC noise analysis or run a group delay analysis.

Linear analysis can be used with all passive circuits and with active circuits that operate under small-signal conditions. In linear analysis the signal level and termination values do not cancel the small-signal conditions, so the superposition principle holds.

After a successful linear-circuit analysis, the results reflect the electrical characteristics of the circuit under small-signal conditions. The results of the analysis are linear network parameters such as scattering (S), admittance (Y), hybrid (H), and impedance (Z), gain parameters, stability parameters, and noise parameters. For details, see Linear Network Analysis in the Circuit help topics.

To add a Nexxim Linear Network Analysis to an HFSS 3D Layout project, right-click **Analysis** in the **Project Manager** window and select **Add LNA Setup** to open the **LNA Solution Setup** 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 add solution options by clicking on “Select” in the "Solution Option” area.. For more information, see Transient Analysis Setup.

Browse the LNA Setup Netlist

The LNA setup is passed to the Nexxim solver via a netlist. To view the Nexxim netlist in the Netlist editor, right-click the LNA setup and select **Browse Netlist** on the menu. For details, see LNA Netlist Syntax in the Circuit help topics.

Set Co-Simulation Options

The Linear Network Analysis runs in co-simulation with a fieldsolver setup in the HFSS 3D Layout project. To select the HFSS Setup for the co-simulation:

1. Click **HFSS 3D Layout** > **Solution Setup** > **Co-Simulation Options** to open the **Co-Simulation Options** window.
2. From the **Solution selection override** area:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Check the **Setup override** area. Select a solution setup from the list from the drop-down in the **Setup override** field.
- Check the **Sweep override** area. Select a sweep from the list on the drop-down in the **Sweep override** field.

3. Select **Renormalize** to re-normalize the Scattering parameter Data (S-data) received on the 3D Layout design before passing it to the Nexxim Circuit solver. Enter the reference impedance value in the ohms box or keep the default of 50 Ohms. If **Renormalize** is not selected, re-normalization does not take place and the original S-data is passed to the Nexxim Circuit solver.
4. Click **OK** to close the **Co-Simulation Options** window.

Run the Linear Network Analysis

Right-click the LNA Setup and select **Analyze** from the menu.

Viewing the Results of the LNA

After the co-simulation has run to completion, you can plot the results of the Linear Network Analysis.

1. Right-click the Reports icon and select **Create Standard Report**>**Rectangular Plot** on the menu.
2. Select your LNA setup as the **Solution**. The **Domain** is set to Sweep. The **Categories** area lists the outputs available on the Linear Network Analysis. Click on a category to view the available **Quantities**. Valid categories are:

- S-parameters
- Y-parameters
- Z-parameters
- ABCD parameters
- H-parameters
- G-parameters
- Gain parameters
- Stability parameters
- Others

For details, see LNA Results in the Circuit help.

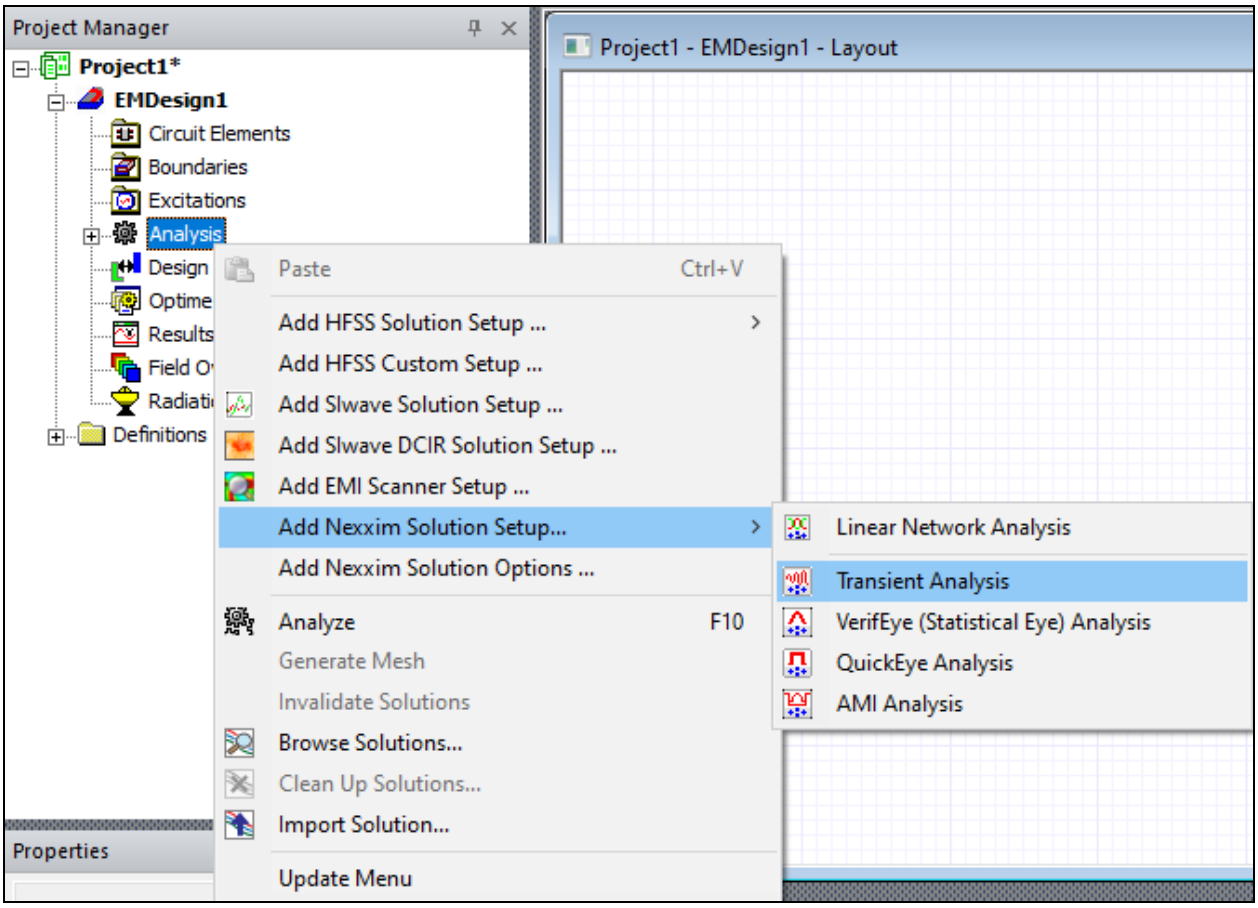
3. Click **New Report** to plot the selected quantities.
4. After LNA has completed, view the solution in the Network Data Explorer. Right-click the LNA setup and select **Network Data Explorer** on the menu. For details, see Network Data Explorer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

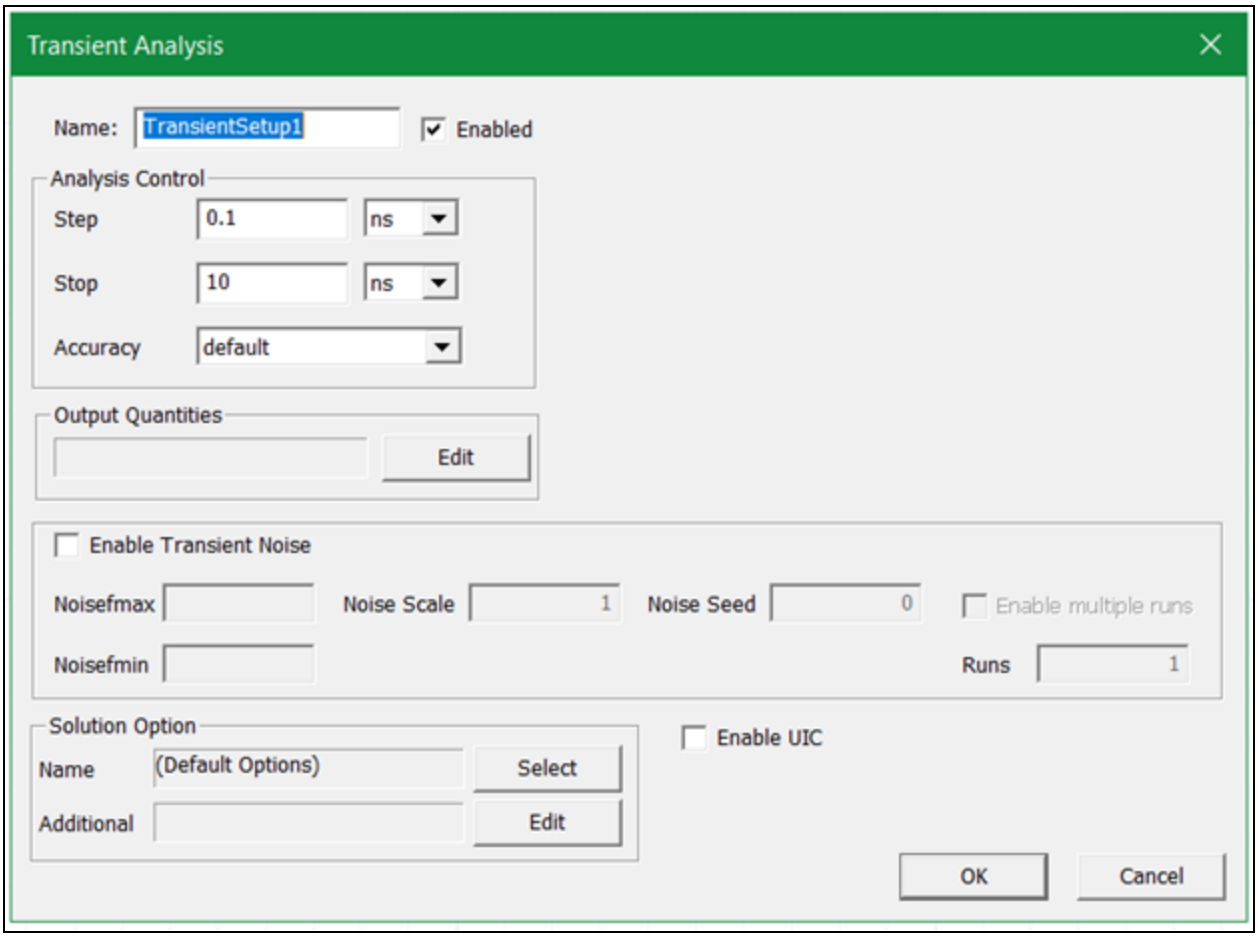
Setting Up a Transient Analysis in HFSS 3D Layout

Follow these steps to setup a Nexxim transient analysis.

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** > **Add Nexxim Solution Setup** > **Transient Analysis** to open the **Transient Analysis** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



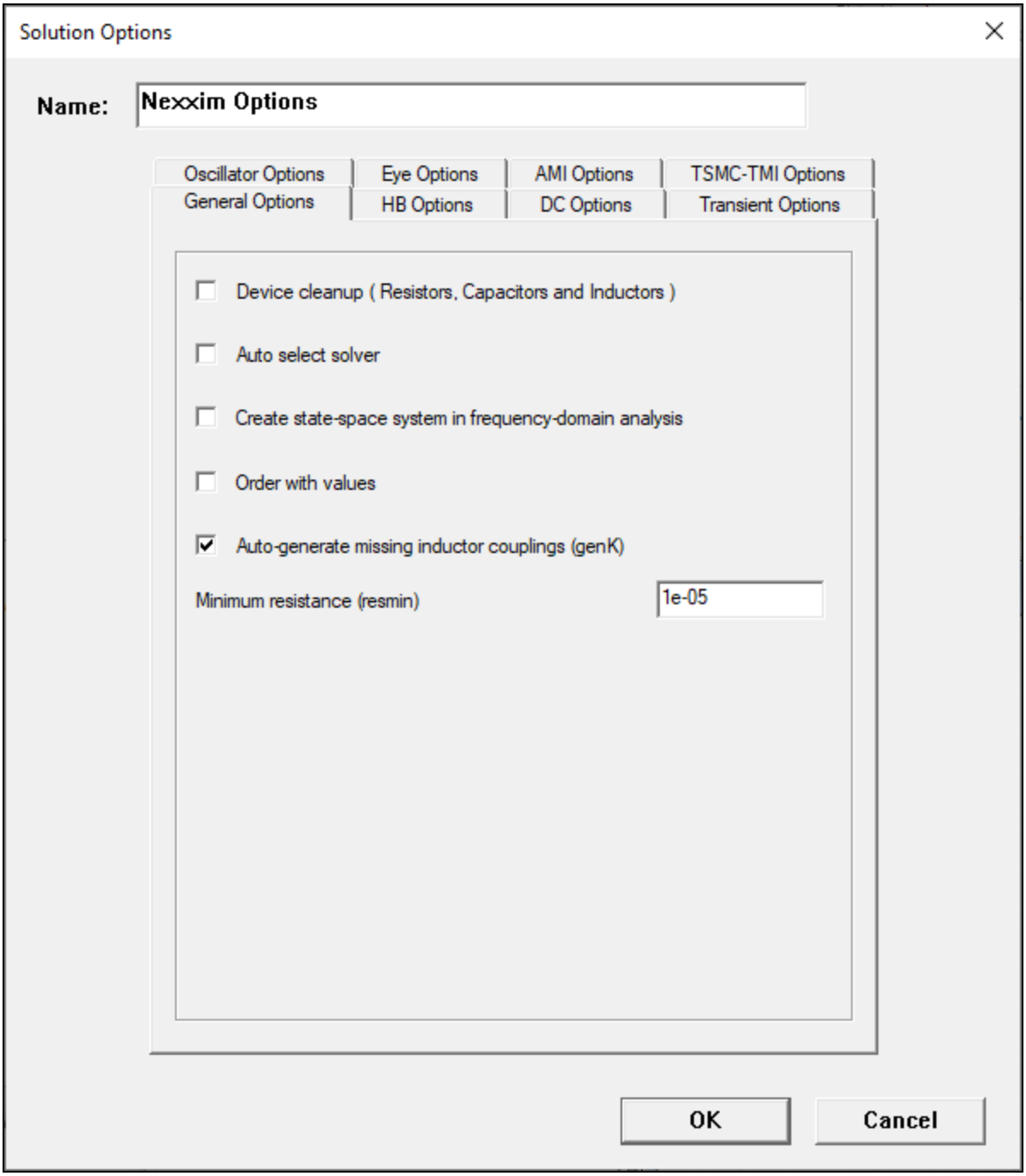
2. To add additional Transient setup options, click **Edit** in the **Solution Option** area to open the **Edit additional options** window. Enter additional options in the field, then click **OK** to

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

close the window.

3. To add additional solution options, return to the active design in the **Project Manager** window and right-click **Analysis**. Then select **Add Nexxim Solution Options** to open the **Solution Options**window and select options on the eight tabs of choices in the window. Click **OK** to close the window.

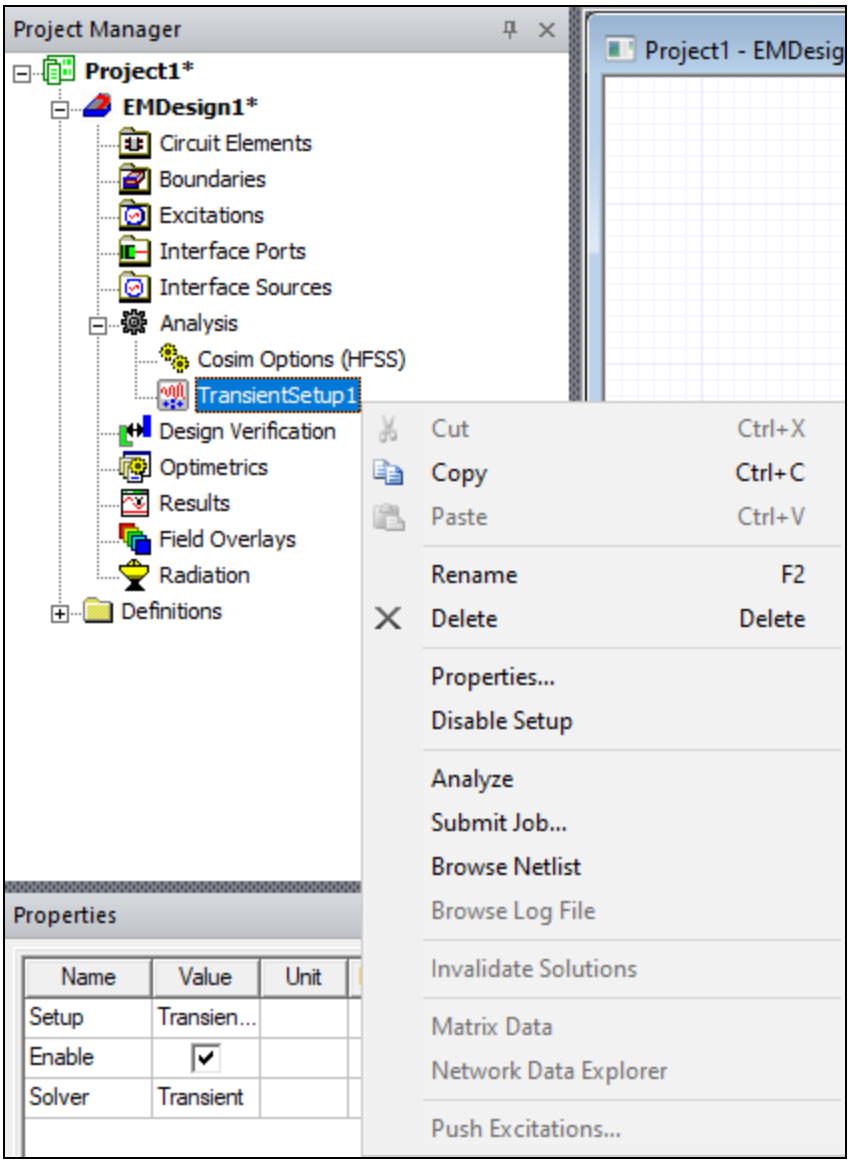
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For more information about the **Transient Analysis** setup window, additional options, and solutions, refer to Running Transient Analysis on the **Schematic Editor**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. After the Transient analysis is set up, a **TransientSetup** icon will appear in the expanded Analysis folder. To netlist the **TransientSetup**, right-click the setup icon and select **Browse netlist**. To start a Nexxim Transient simulation, right click the setup icon and select **Analyze**.



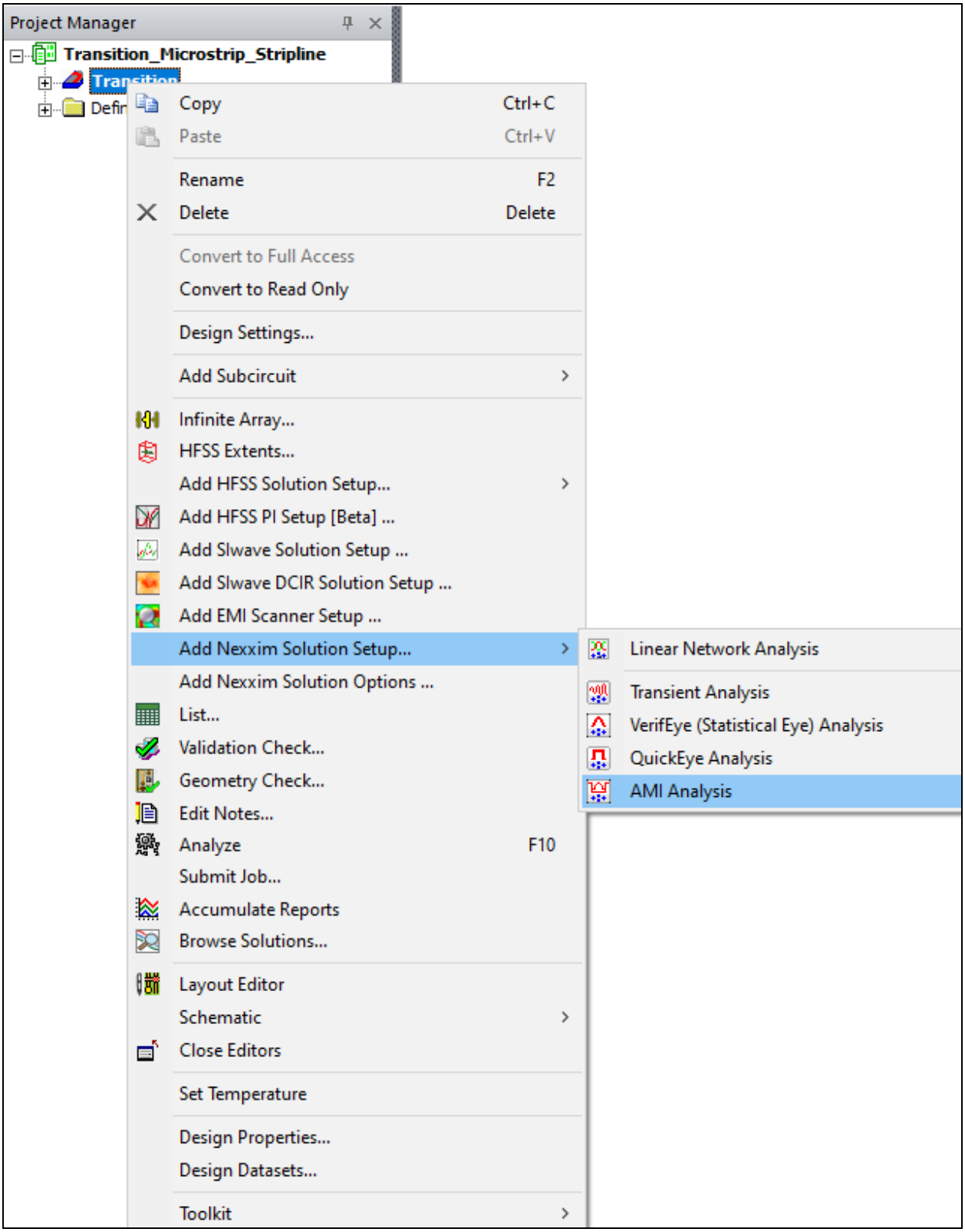
AMI Analysis in HFSS 3D Layout

From the **Schematic Editor**, perform these steps to set up, run, and view the results of an AMI analysis in HFSS 3D Layout. For additional information, see AMI 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. From the **Project Manager** window, expand the **Project Tree** and right-click **[*Active Design Folder*]**. Then select **Add Nexxim Solution Setup** > **AMI Analysis** to open the **AMI Analysis** window.

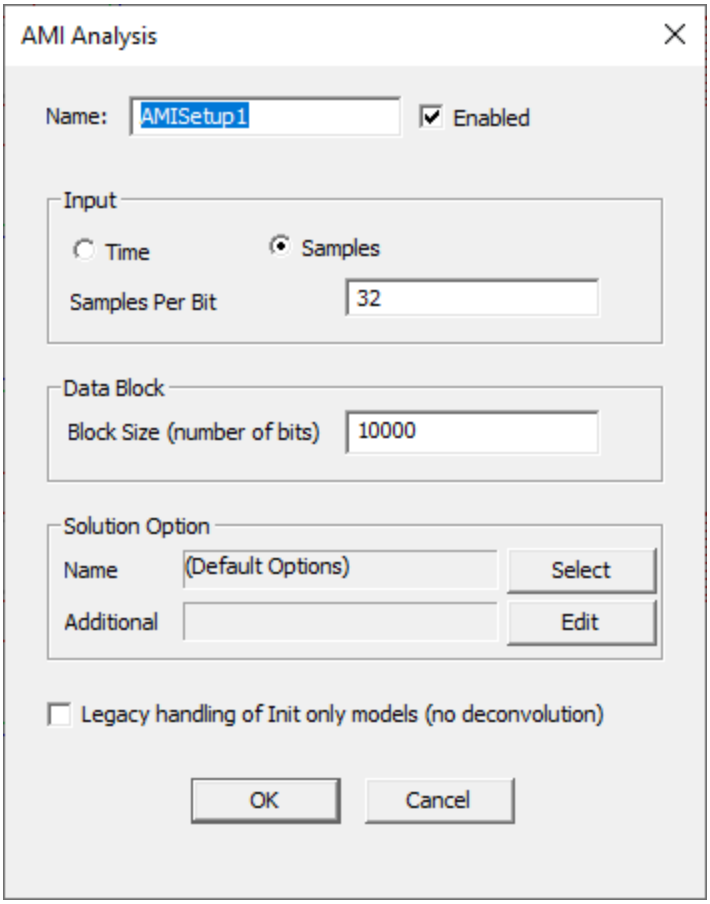
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Alternatively, on the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** and select **Add Nexxim Solution Setup** > **AMI Analysis**.



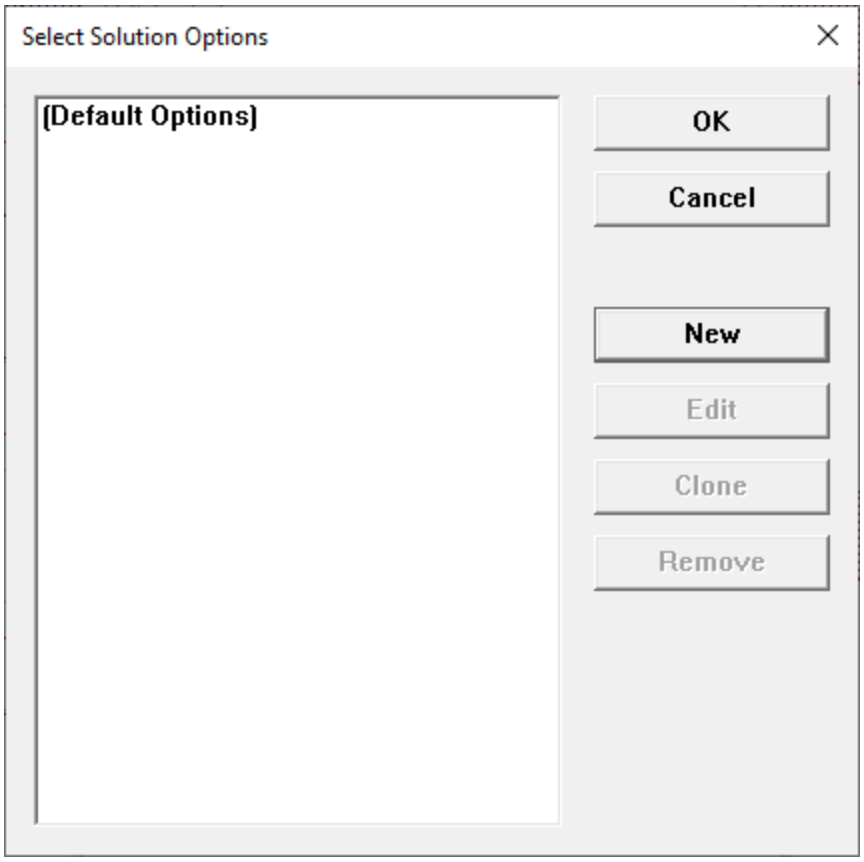
2. Ensure the **Enabled** box is checked (default setting).
3. AMI simulation uses a fixed timestep for sampling the impulse response and any processed waveforms. From the **Input** area, choose **Time** or **Samples** to define the unit interval (UI) at which the impulse response and waveforms are sampled.

- Use **Samples** to specify the number of samples per bit (default is 32 samples per bit). Most AMI models and simulations are set up using **Samples** per bit.
- Use **Time** to set the absolute timestep in seconds. The timestep must be greater than 0, and must be a fraction of the UI. The default timestep is 5e-10 seconds.

6. The **Block Size (number of bits)** field sets the number of bits per AMI data block (default is 10,000 bits per block).

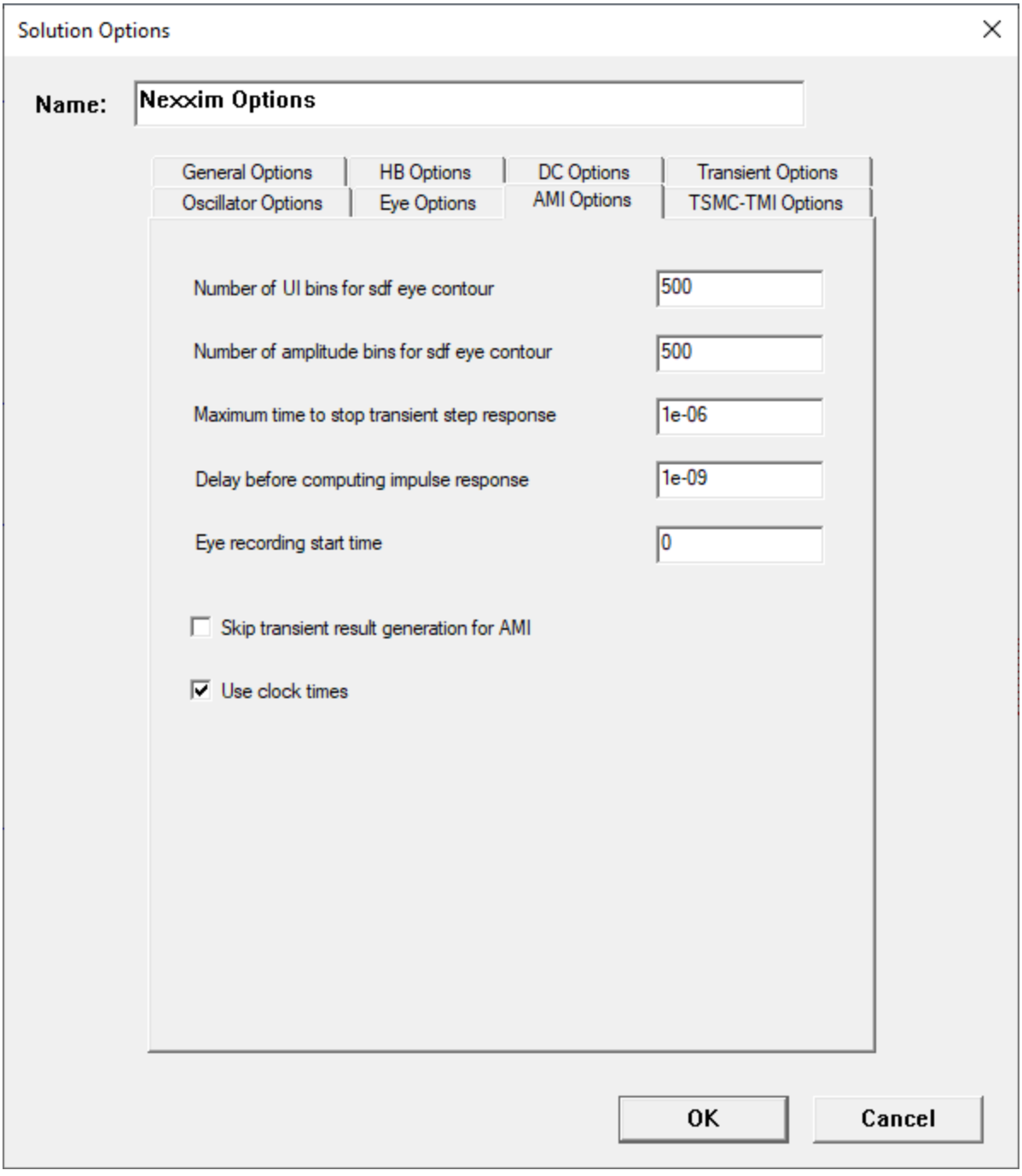
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Optionally, use the fields in the **Solution Option** area to add AMI options.
   - Click **Select** to open the **Select Solution 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.

- To create a new option set, click **New** to open the **Solution Options** window.



- Enter a for the new option set in the **Name** field.
- Choose on the fields and check boxes in the *AMI Options* to set options, as appropriate.

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, open the **Solution Options** window on the **Project Manager**. Either expand the **Project Tree** and right-click **[*Active Design Folder*]**, or expand **[*Active Design Folder*]** and right-click **Analysis**. Then select **Add Nexxim Solution Options** .

Option	Default Value	Description
ami.ampl_bins	500	Number of histogram bins to use for generating the 3D and contour data.
ami.delay	1	Delay before computing impulse response.
ami.eye_start_time	0	Start time for recording eye data.
ami.resp_tmax	1e-6	Maximum time for the transient impulse response calculation.
ami.skip_tran_result	0	0=Save transient data for AMI 1=Skip transient result generation for AMI (useful for long runs of data that need not be saved).
ami.ui_bins	500	Number of histogram bins to use for generating the 3D and contour data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ami.use_clock_times	1	1=Turn on sampling of the eye diagram based on the clock time values produced by the AMI receiver model. 0=Do not use clock times on the AMI receiver. The eye diagram are sampled at even UI intervals. The results of clock time sampling can be observed only in the Statistical Eye Plot for the "EyeAfterProbe" quantity.

Eye Sources and Probes in HFSS 3D Layout

Eye probe and eye source components can be used in 3D Layout. Their usage is described in the following help topics.

QuickEye and VerifEye Setup in HFSS 3D Layout

The eye diagram is a convenient way to analyze the performance of a serial communications channel. In a traditional eye diagram, copies of the waveform generated by transient analysis are overlaid at a spacing of one unit interval (UI). The width of the eye is affected by timing variations such as jitter and by variations in setup and hold times.

For additional information, see the following topics in Circuit Time Domain Analyses:

VerifEye And Quick Eye Analyses

VerifEye and Quick Eye Analysis Options

Running VerifEye Analysis on the **Schematic Editor**

Running Quick Eye Analysis on the **Schematic Editor**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

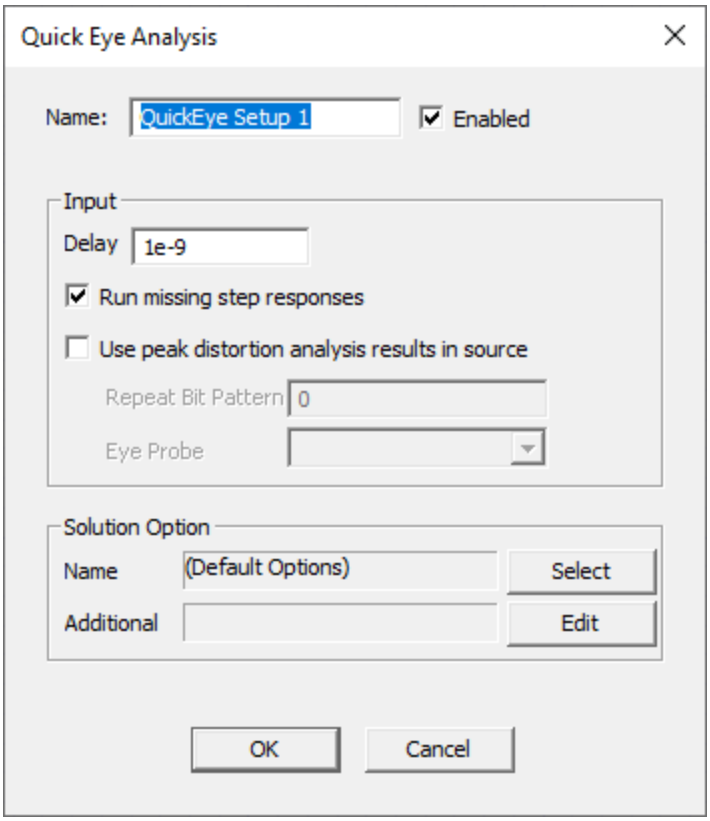
Resolutions of Eye Diagrams

QuickEye and VerifEye Technical Notes

QuickEye and VerifEye References

Adding QuickEye Setup in HFSS 3D Layout

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** and select **Add Nexxim Solution Setup** > **Quick Eye Analysis** to open the **Quick Eye Analysis** window.



2. Type a **Name** (or accept the default name, e.g., “QuickEye Setup 1”).
3. Leave the **Enabled** area selected (default setting).
4. Set the **Delay** parameter to control the delay in applying the step response.
5. **Run missing step response** allows simulation to proceed when an External Step Response component is present in the design but no step response data file can be found.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

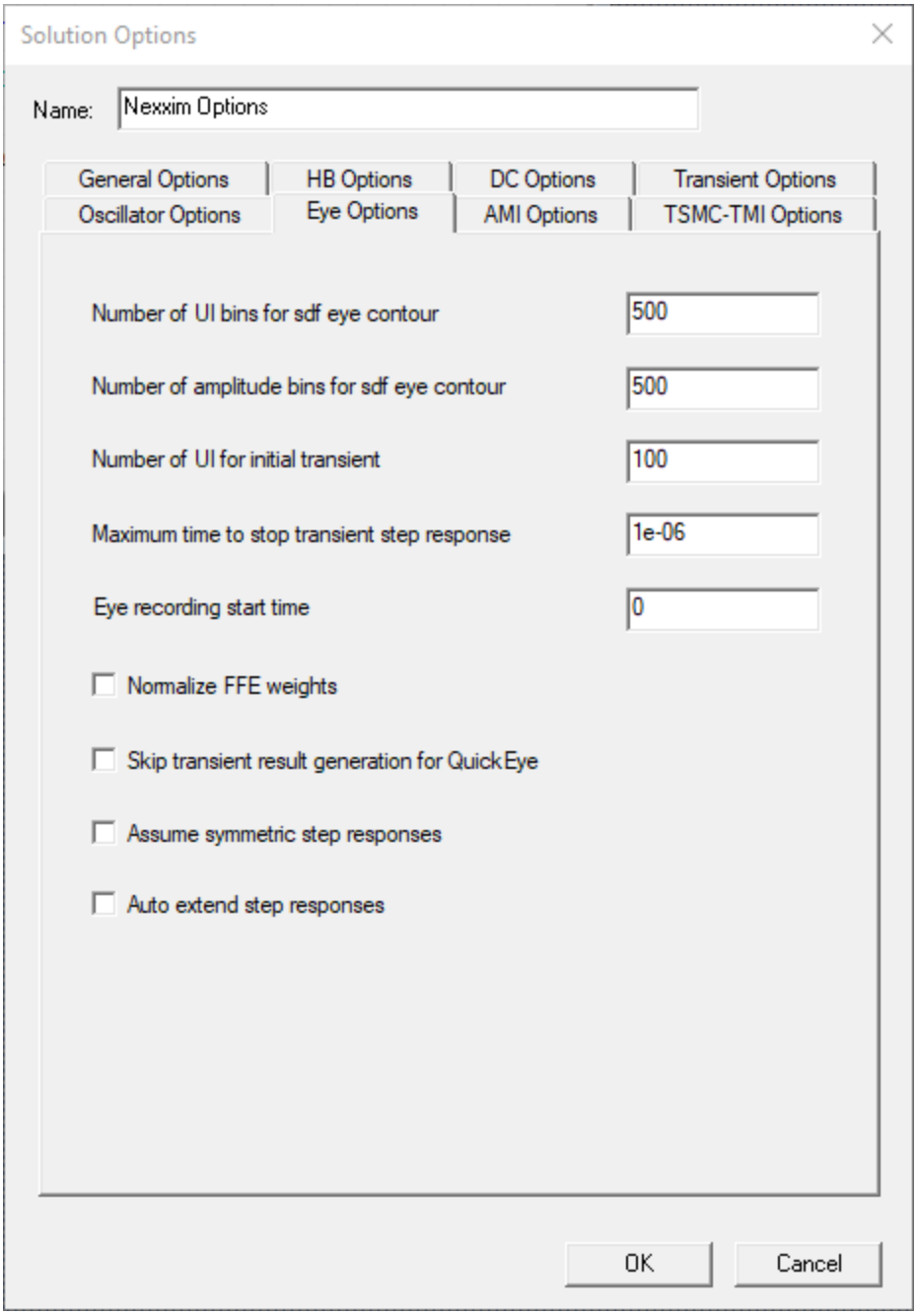
6. Click the **Use peak distortion analysis results in source** check box to enable the worst-case bit pattern that could be output on the source. Set the **Repeat Bit Pattern** field to the selected number of repeats. When the **Use peak distortion** box is checked, the worst-case bits from Peak Distortion Analysis are output from all Eye Sources. Any bit patterns set in the sources is ignored. See Peak Distortion Analysis for details.

- Specify the number of times to repeat the worst-case bit pattern
- Select an Eye Probe to target with all Eye Sources. Optionally, use the fields in the **Solution Option** area to add **VerifEye/QuickEye** options.
- Click **Select** on the **Name** field to open the **Select Solution Options** window.
- To create a new option set, click **New** to open the **Solution Options** window.
- Click the **Name** field to name the new option set.
- Click the **Eye Options** area fields and check boxes to set any selected options.

Nexxim Solution Options for QuickEye

From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** and select **Add Nexxim Solution Options** to open the **Solution Options** window. Select the **Eye Options** tab.

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 UI bins** sets the number of UI histogram bins to use for generating the 3D and contour data. The default is 500. Option: **eye.ui_bins**.
- **Number of amplitude bins** sets the number of amplitude histogram bins to use for generating the 3D and contour data. The default is 500. Option: **eye.ampl_bins**.

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 UI** sets the number of unit intervals (UI) over which the initial transient step response should run after any channel or source delay. The default is 100. Option: **eye.num_initial_ui**.
- **Maximum time** sets the maximum stop time for the transient step response calculation. The default is 1❍s. Option: **eye.resp_tmax**.

  > **Note:**
  >
  > QuickEye uses the minimum of (num_initial_ui x UI) and (resp_tmax) to set the transient final time for the step response calculation.

  Normalize FFE weights sets the solver to normalize FFE weights (sum of absolute values of all weights equals one). This option applies only to weights supplied by users. The Circuit solver always normalizes the taps weights that it calculates. The default for user-supplied weights is no normalization. Option: **eye.normalize_ffe_weights**.
- **Skip transient result** turns off QuickEye storage of the eye diagram and the transient-like result. By default, QuickEye stores the eye diagram and the transient-like result. Option: **eye.qe_only_cmf**.
- **Assume symmetric step responses** sets QE/VE to use the rise time as the fall time. The default is to use different rise and fall times. Option: **eye.sym_step_resp**.
- **Auto extend step responses** extends the step response beyond the simulation time. By default, the step response is extended only to the end of simulation time. Option: **eye.auto_extend_step_resp**.

For more information, see Circuit Time Domain Analyses: VerifEye and Quick Eye Analysis Options.

Quick Eye Analysis Outputs

For information about Quick Eye analysis outputs in HFSS 3D Layout, refer to Circuit Time Domain Analyses: Display Quick Eye Analysis Outputs.

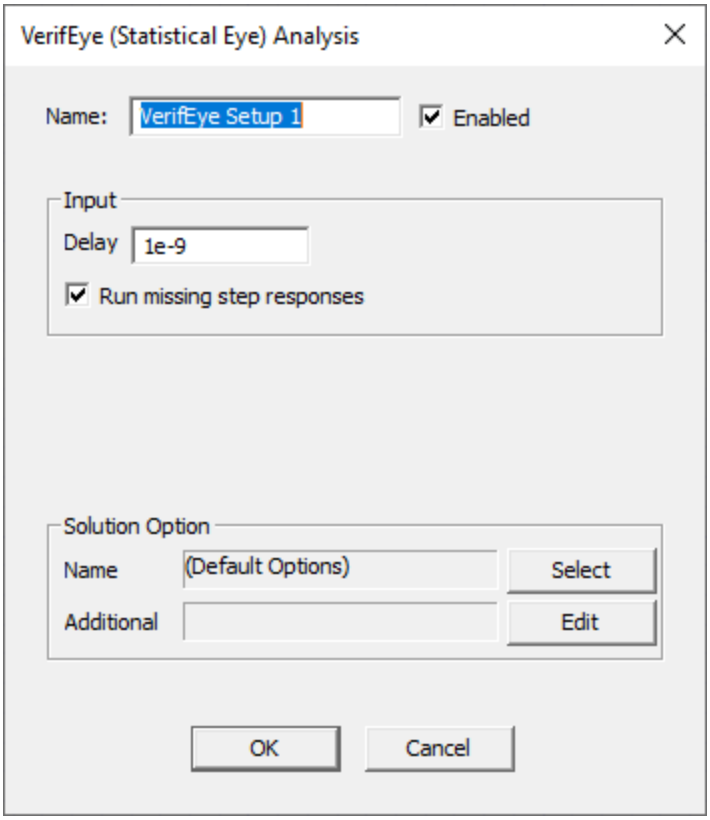
Run the Quick Eye Analysis and Display Results

For information about running the Quick Eye analysis in HFSS 3D Layout, refer to Circuit Time Domain Analyses: Run the Quick Eye Analysis and Display Results.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding VerifEye Setup in HFSS 3D Layout

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** and select **Add Nexxim Solution Setup** > **VerifEye Analysis** to open the **VerifEye Analysis** window.



2. Type a **Name** (or accept the default name, e.g., “VerifEye Setup 1”).
3. Leave the **Enabled** area selected (default setting).
4. Specify a **Delay** to apply before the step response is applied (default is 0 seconds).
5. **Run missing step response** allows simulation to proceed when an External Step Response component is present in the design but no step response data file can be found.

VerifEye Analysis Outputs

For information about VerifEye analysis outputs, refer to Circuit Time Domain Analysis: Display VerifEye Analysis Outputs.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

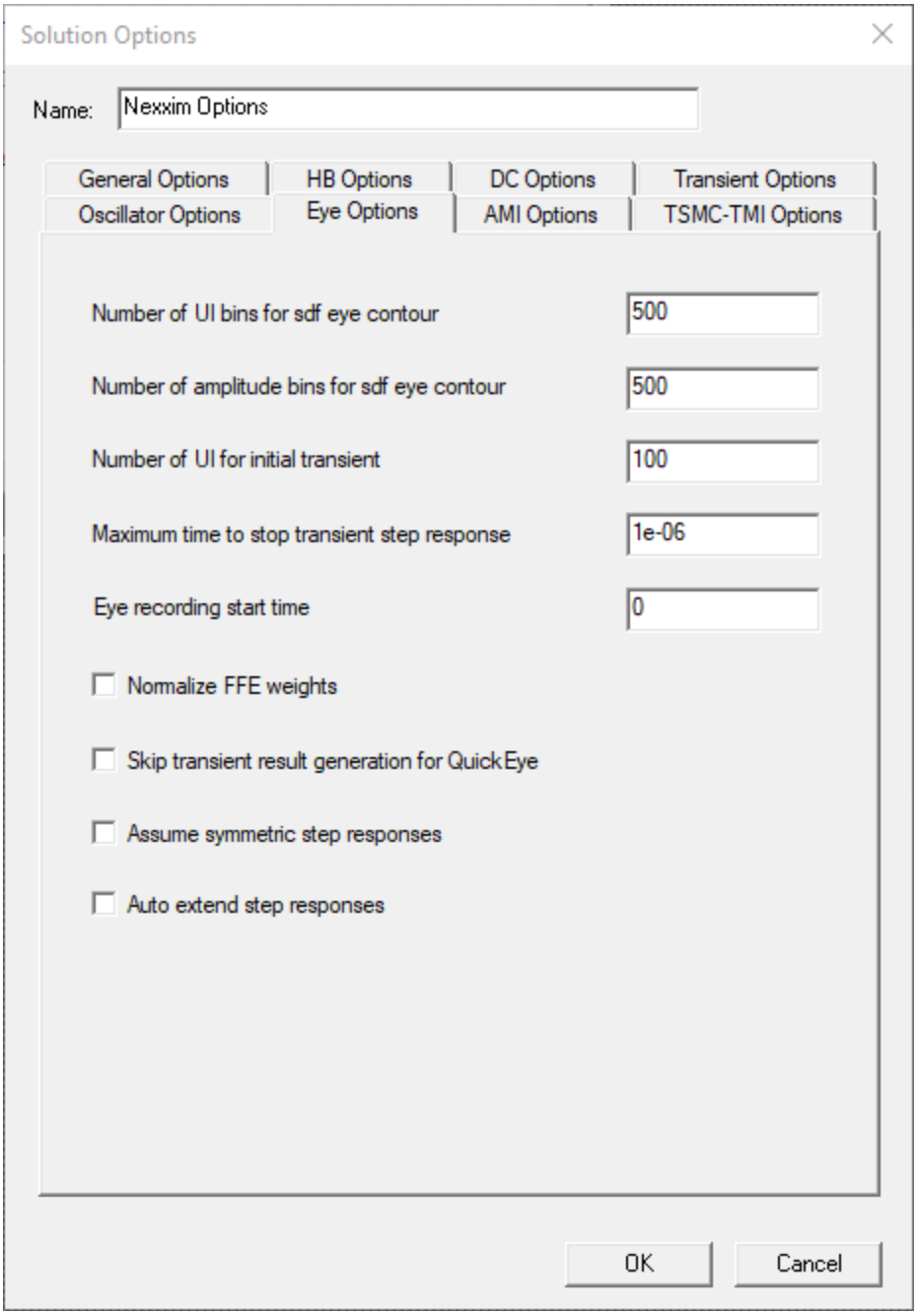
Run the VerifEye Analysis and Display Results

For information about running VerifEye analysis, refer to Circuit Time Domain Analyses: Run the VerifEye Analysis and Display Results.

Nexxim Solution Options for VerifEye

From the **Project Manager** window, right-click **Analysis** and select **Add Nexxim Solution Options** to open the **Solution Options** window. Select the **Eye Options** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Skip transient result** turns off QuickEye storage of the eye diagram and the transient-like result. By default, QuickEye stores the eye diagram and the transient-like result. Option: **eye.qe_only_cmf**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Assume symmetric step responses** sets QE/VE to use the rise time as the fall time. The default is to use different rise and fall times. Option: **eye.sym_step_resp**.
- **Auto extend step responses** extends the step response beyond the simulation time. By default, the step response is extended only to the end of simulation time. Option: **eye.auto_extend_step_resp**.

For more information, see Circuit Time Domain Analyses: VerifEye and Quick Eye Analysis Options.

Subcircuit Optimization in HFSS 3D Layout and Nexxim

HFSS 3D Layout and Nexxim both support matrix quantities in optimetrics (Optimization, Parametic Analysis, etc). From the Setup Optimization window, users can select matrix quantities like S-Matrix, Y-Matrix, and Z-Matrix as calculations, and can then specify as a goal an equivalent matrix or an imported solution of the same matrix size.

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform provides all the S-Matrix, Y-Matrix and Z-Matrix data for optimization. If you select an imported solution as the goal, the S-Matrix data on the imported solution is read in and sent to the optimization routines. For more information, see Setting up an Optimization Analysis.

Adding a Frequency Sweep (HFSS 3D Layout)

For Driven solution types.

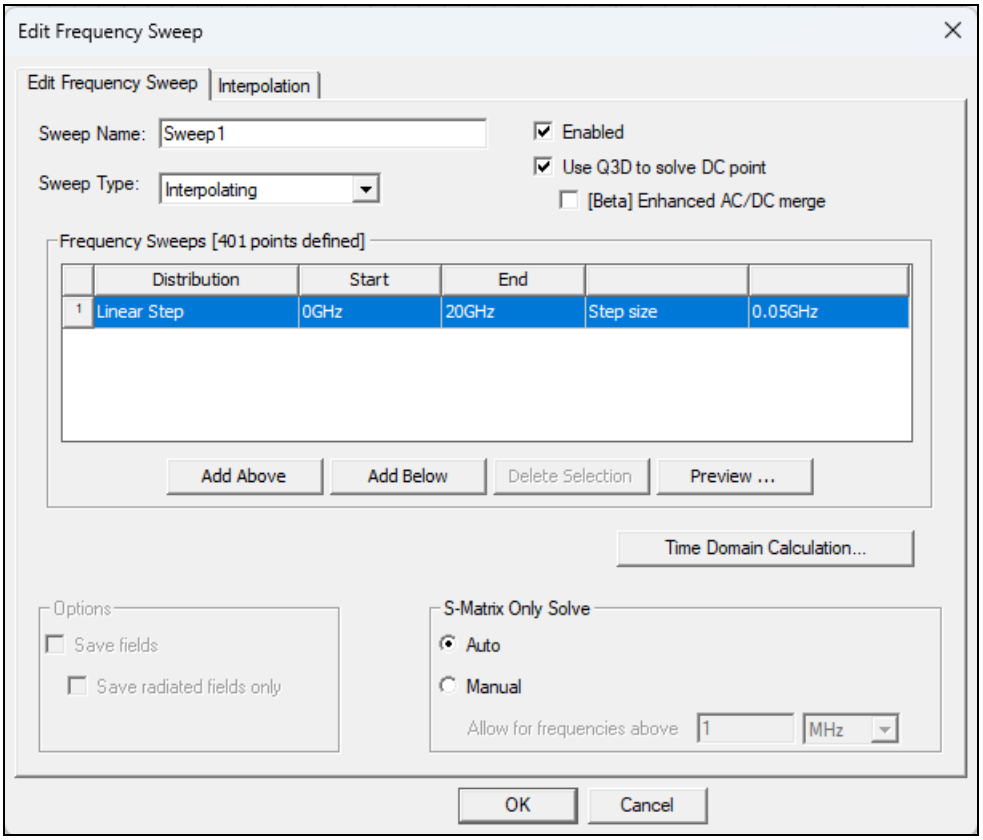
To generate a solution across a range of frequencies, add frequency sweeps to solution setups. HFSS 3D Layout performs the sweep after the adaptive solution. If an adaptive solution is not requested, the sweep is the only solution generated. A sweep can also be deactivated, so only the adaptive solution is run (or a ports-only solution is run), then later reactivated. Many of the frequency settings are the same in HFSS 3D Layout and HFSS.

Follow these steps to add a frequency sweep to a solution setup.

1. From the **Project Manager** window, expand the **Project Tree** and select the HFSS 3D Layout design to make the **HFSS 3D Layout** menu appear.
2. From the **HFSS 3D Layout** menu, select **Solution Setup** > **Add Frequency Sweep** to

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

open the **Edit Frequency Sweep** window.



3. Specify the following frequency sweep parameters:

- **Sweep Name** field – enter a name for the sweep.
- **Sweep Type** drop-down menu – select **Interpolating** (default), **Discrete**, or **Broadband Fast**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> HFSS 3D Layout supports **only Interpolating**, **Discrete**, and **Broadband Fast** frequency weeps. HFSS 3D Layout does **not** support:
>
> - Minimum solutions
> - Minimum number of sub ranges
> - Network analysis 3D solution interpolating sweeps (Driven modal)
> - Network analysis ports-only, interpolating sweeps (Driven modal)
> - Network analysis ports-only, interpolating (Driven terminal)
> - Derivatives
> - Set defaults option for error tolerance and max solutions
> - Mode selection for interpolation basis convergence

- **Enabled** check box – check to enable the sweep.
- **Use Q3D to solve DC point** check box – check to invoke the Q3D Extractor to explicitly solve for a DC point. The mesh from HFSS is used for the Q3D extraction, with sources and sinks defined based on HFSS port assignment. Matrix reductions are automated to integrate the Q3D results with HFSS.

> **Note:**
>
> **Use Q3D to solve DC point** provides network data only and cannot be used for fields overlays.

  - **Enhanced AC/DC merge** check box – check to enforce causality and generate S-parameters at the DC point. By combining the DC results with the frequency-swept AC results, accuracy is improved over the entire simulation bandwidth.

    > **Note:**
    >
    > **Enhanced AC/DC merge check** is only available if the **Sweep Type** is **Interpolating** and the **Enhanced low frequency accuracy** box is checked on the **Solver** tab during setup.

- ***Frequency Sweeps*** area: specify sweeps in terms of **Distribution** type (i.e., **Single Point**, **Linear Step**, **Linear Count**, or **Log Scale**).

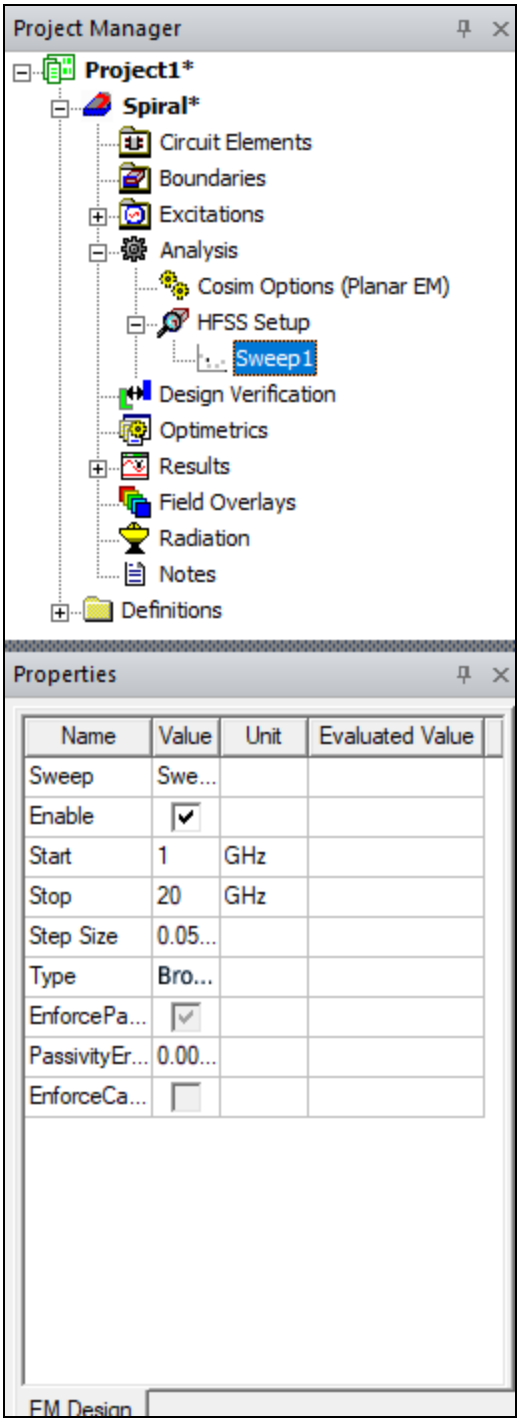
4. If applicable, click **Add Above** or **Add Below** to add additional frequency sweeps, including mixed sweep types. This feature provides flexibility (e.g., define sweeps with log scale at lower frequencies, and linear step at higher frequencies).

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 applicable, click **Delete** to remove any sweeps.
6. Click **Preview** to display the sweep(s) as currently defined.
7. Click **OK** and the sweep(s) will appear in the **Project Manager** window > **Project Tree** > **[*Active Design Folder*]** > **Analysis** directory, under the associated setup and with the **Sweep Name** applicable, in **Step 3**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Select the **Sweep Name** to populate the **Properties** window with related parameters.



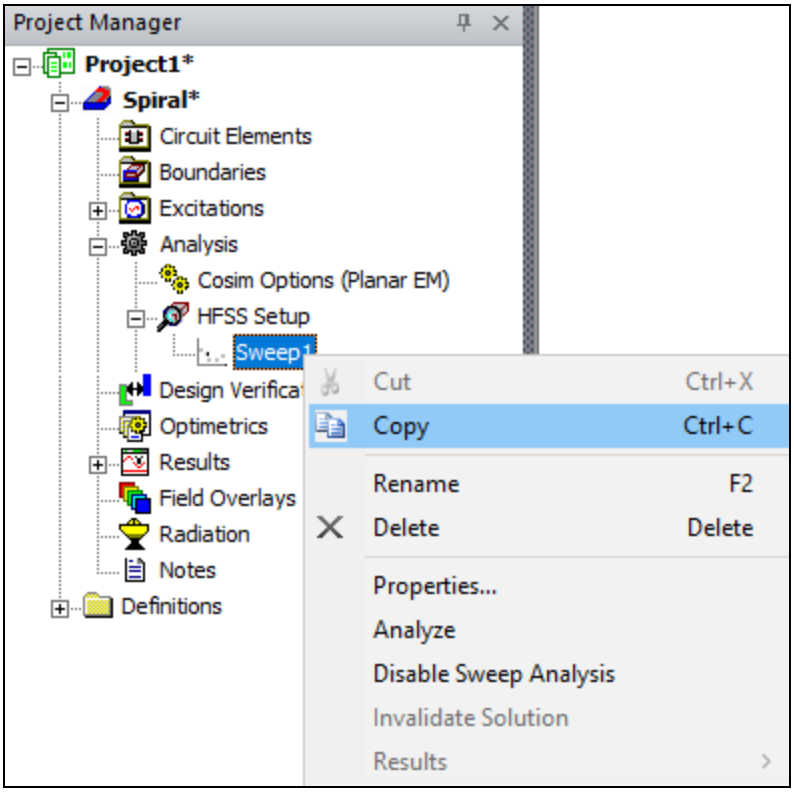
The procedure is complete.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy, Paste, and Edit Frequency Sweeps

Follow these steps to copy, paste, and/or edit a frequency sweep.

1. From the **Project Manager** window, expand the **Project Tree** > **[*Active Design Folder*]** > **Analysis** > **Setup**.
2. Right-click a sweep and select **Copy**.

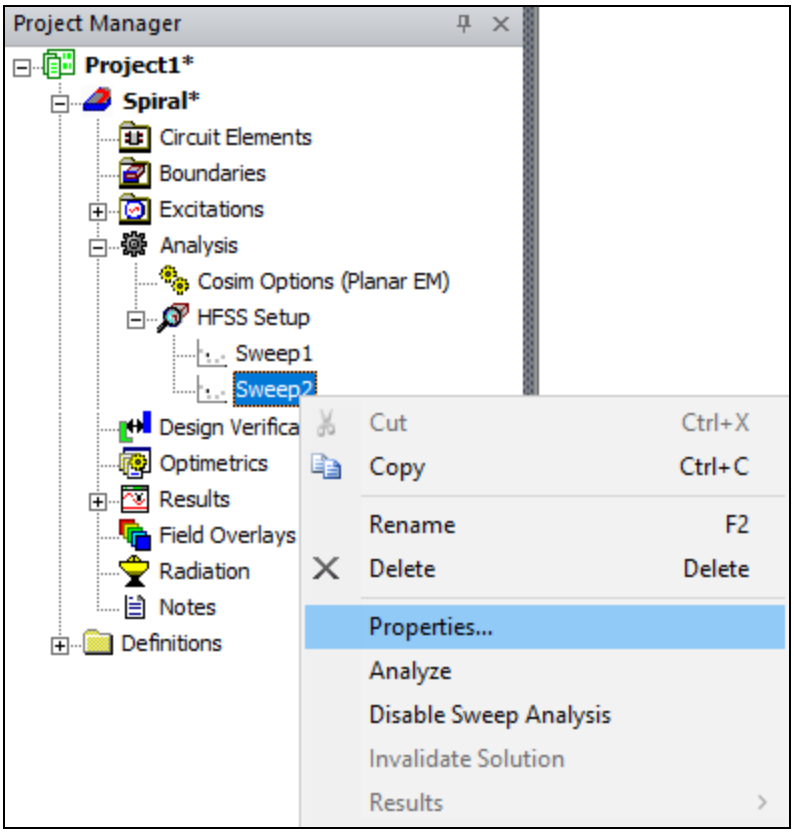


3. Right-click a setup and select **Paste** to create a copy of the sweep. By default, the copy is named **Sweep*n***.

> **Note:**
>
> The **Paste** command for frequency sweeps is design sensitive. There is no way to paste between Driven and Eigenmode designs and sweeps can only be pasted in a setup. Dependent setups are pasted along with setup copies. A warning will appear if the dependent setup is already in the design and setup is not pasted again.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Right-click the **Sweep*n*** copy and select **Properties** to reopen the **Edit Frequency Setup** window and make changes.



Selecting the Sweep Type

1. From the **Project Manager** window, expand the **Project Tree** and select the active design to make the **HFSS 3D Layout** menu appear.
2. From the **HFSS 3D Layout** menu, select **Solution Setup** > **Add Frequency Sweep** to open the **Edit Frequency Sweep** window.
3. Select one of the following sweep types:

Interpolating Sweep Options (Default)	Generates a solution for an entire frequency range. Best for models when the frequency range is wide and the frequency response is smooth, or if the memory requirements of a fast sweep exceed available resources. All discrete basis solutions are solved prior to interpolating sweeps because it is possible that an interpolating sweep can re-use already solved frequencies from a discrete sweep.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	An interpolating sweep must be used for Time Domain Reflectometry plots (TDR). When you select interpolating sweep, the **Options** tab is activated. This lets you specify a maximum number of solutions, and other interpolation values.
Discrete Sweep Options	Generates field solutions at specific frequency points in a frequency range. Best when only a few frequency points are appropriate to accurately represent the results in a frequency range.
"Options for Broadband Fast Sweeps" below	Generates a unique full-field solution for each division within a frequency range. Best for moderately-sized models when the frequency range is wide and the frequency response is complicated, where interpolating or discrete sweeps would require too many specific frequency points. When you select Broadband Fast Sweep, the **Options** tab is activated. This lets you specify a maximum number of solutions and error tolerance.

Options for Broadband Fast Sweeps

> **Note:**
>
> The Broadband Fast Sweep uses the same lumped port model as the **Enhanced low frequency accuracy** option available in the **HFSS Setup** window to all frequency sweeps. This is the case whether the **Enhanced low frequency accuracy** check box is checked or otherwise.

The **Edit Frequency Sweep** window for broadband fast sweeps includes the following options:

Edit Frequency Sweep Tab

- **Sweep Name**: enter a name for the sweep in the field.
- **Enabled** check box: active or deactivate the sweep.
- ***Frequency Sweeps*** area .
- **Use Q3D to solve DC Point** check box: choose whether to use Q3D to solve DC point.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

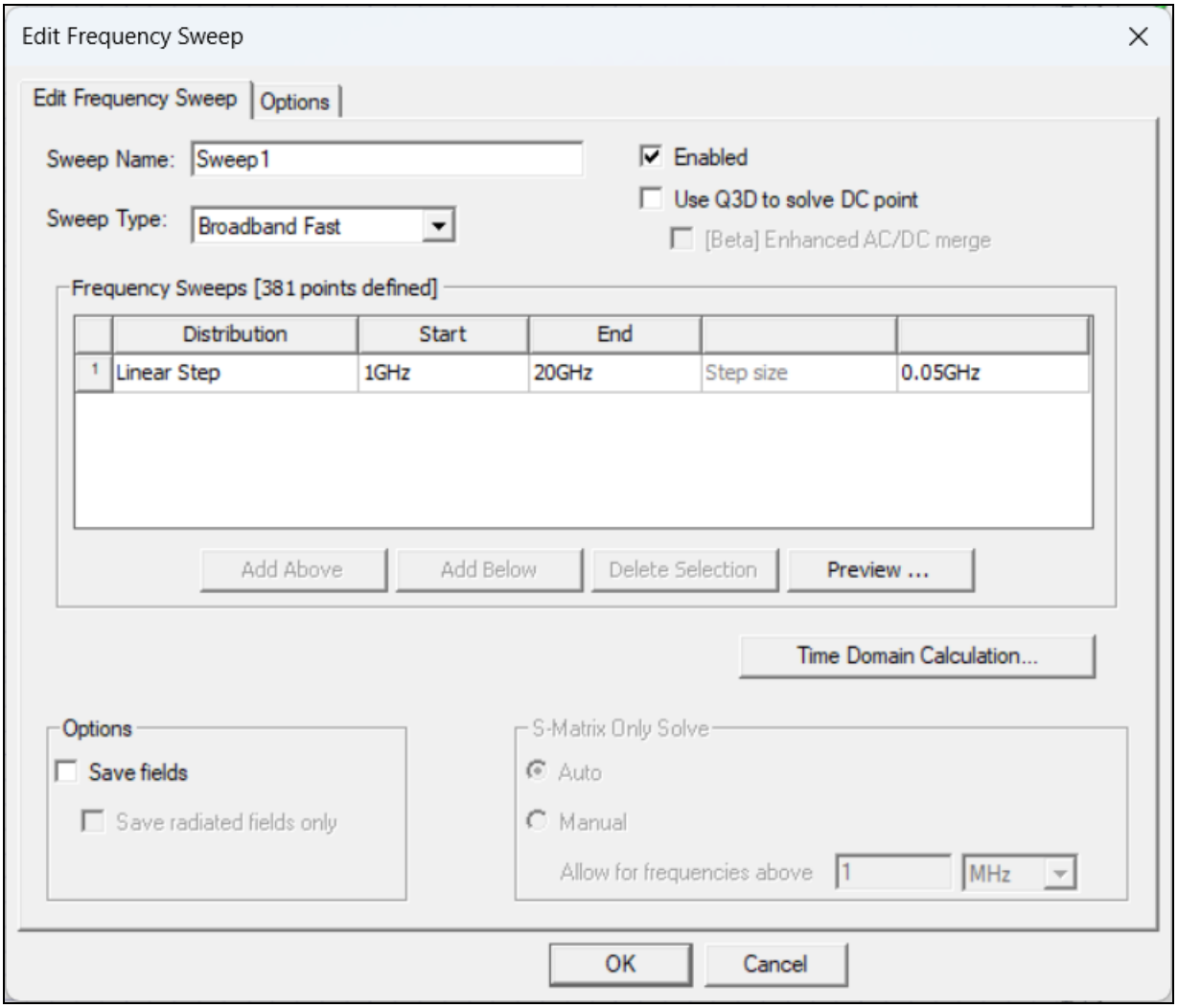
Use Q3D to solve DC point provides network data only and cannot be used for fields overlays.

- **Options** area: choose whether to check the **Save Fields** box and/or the **Save radiated fields only** (default is unchecked). If the **Save Fields** box is checked, HFSS 3D Layout solves the Broadband Fast Sweep, computes the fields at each frequency in the sweep, then saves them. This has two effects: the simulation takes longer, but post-processing is much faster. Since these options are exercised at solve time, they do not apply to existing solutions. Enabling this option for a previously solved sweep and re-solving will access the previously solved data and generate the requested fields. If there are more than one hundred frequencies, checking the **Save Fields** box generates a warning that disk space use may be excessive.

Note:

Selecting **Save Fields** enables the check box for **Save radiated fields only**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



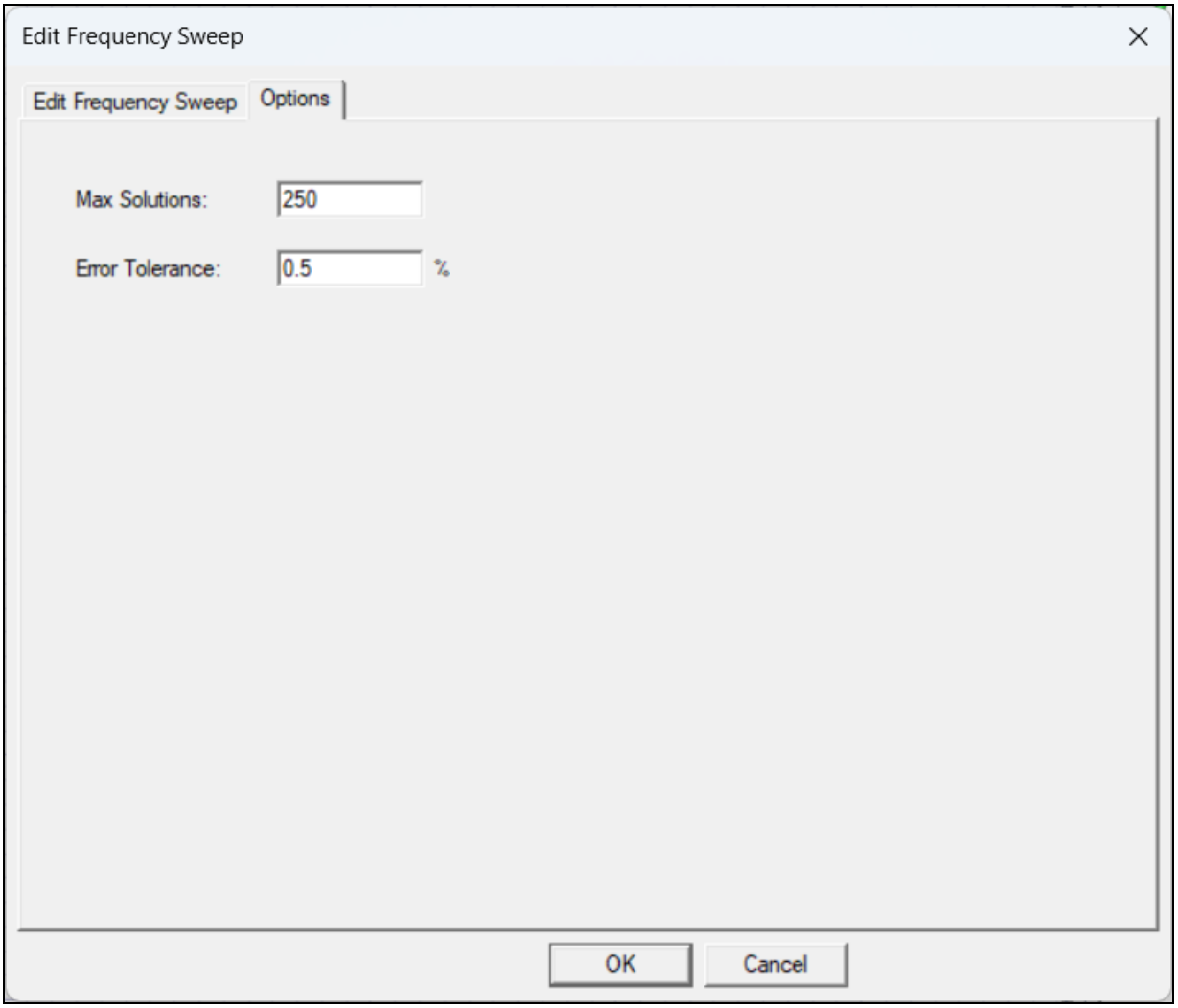
Options Tab

From the **Options** tab, do the following:

Max Solutions: enter the maximum number of solutions that can be solved for the frequency range (default is 250). The value must be greater than 3.

Error Tolerance: set the maximum relative difference allowed between two successive points (default is 0.5). The default set the error tolerance percentage.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Choosing Frequencies for Full-Wave SPICE

If you plan to perform a full-wave SPICE analysis, use the **Time Domain Calculation** dialog box to help determine a suitable frequency sweep range for the solution.

To perform the calculation of suitable frequencies to solve:

1. In the **Edit Sweep** dialog box, click **Time DomainCalculation**.

   The **Time Domain Calculation** dialog box appears.

2. Type a minimum rise time value in the **Signal Rise Time** box.

   This value represents the time scale that will characterize the rate of change of the input time signal, which will be applied in the circuit simulator.

3. Type a value in the **Time Steps Per Rise Time** 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 time sampling increment for the entire signal is calculated using

$$\Delta t = \frac{\tau}{N_\tau} \quad (1)$$

where

- Δt is the time sampling increment.
- τ is the signal rise time.
- N_τ is the number of time steps per signal rise time.

4. Type a value in the **Number of Time Points** box.

Note that the input time signal duration is determined using $N \times \Delta t$, where N is the number of time points.

5. Click **Calculate**.

- HFSS now determines the **Maximum Frequency** using

$$F_{max} = \frac{0.5}{\Delta t} \quad (2)$$

- where F_{max} is the maximum frequency.

HFSS determines the **Frequency Step Size** using $\frac{F_{max}}{N}$.

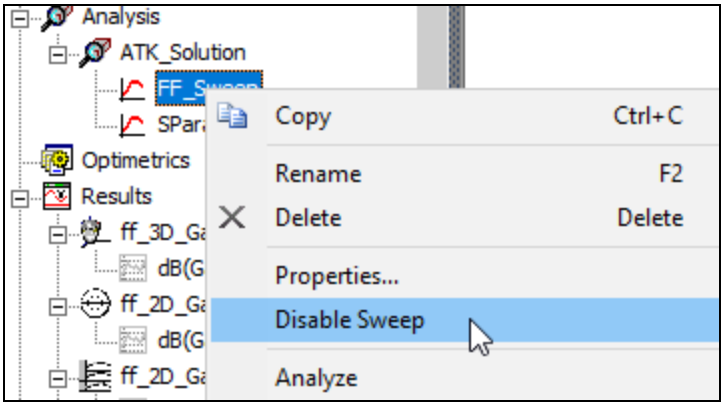
6. Click **OK** to transfer the data to the frequency sweep fields in the **Edit Sweep** dialog box.

Disabling or Enabling a Frequency Sweep

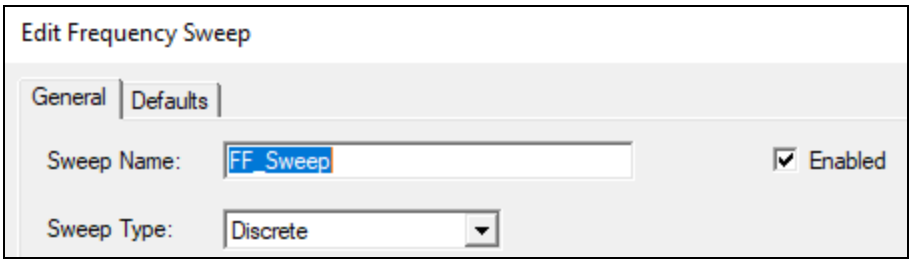
You can activate or deactivate a sweep definition. Disabled sweeps has an “empty graph” icon, and are not analyzed until enabled. To deactivate a sweep definition without deleting it:

1. Expand the tree hierarchy under the **Analysis** icon in the **Project Tree**.
2. Expand the tree hierarchy under the icon for the analysis setup that includes the sweep.
3. Right-click the icon for the sweep definition and select **Disable Sweep Analysis**. (To reactivate the sweep, right-click again and select **Enable Sweep Analysis**.)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Also deactivate a sweep by right-clicking and selecting **Properties** to open the **Edit Properties** window. In the upper-right corner of the **Edit Properties** window, uncheck the **Enabled** area. Then click OK to apply the change. (To reactivate the sweep, re-open the **Edit Properties** window and check the **Enabled** area. Then click OK.)



Q3D-Based Electrothermal Workflow in HFSS 3D Layout (Harmonic Loss)

Background

This workflow addresses power PCBs and packages for power electronics converters. In most cases, these operate under Pulse Width Modulation (PWM), resulting in voltages and currents with rich harmonic content.

In these cases, it is difficult to define DC equivalent currents that:

- Accurately represent power losses, and
- Satisfy the current balance in accordance with Kirchhoff's first law.

Therefore, an advanced workflow is required to accurately estimate losses in bus bars, connectors, and PCBs for power electronics.

The workflow includes the following tools:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Q3D solvers in HFSS 3D Layout – used to extract parasitics of a PCB or package
- Circuit with a Transient Nexxim setup – used to dynamically link parasitics model from Q3D solvers in HFSS 3D Layout
- SIwave DCIR solver in HFSS 3D Layout with pull excitations from Circuit – used for harmonic loss calculation

Workflow using Q3D Extractor in HFSS 3D Layout

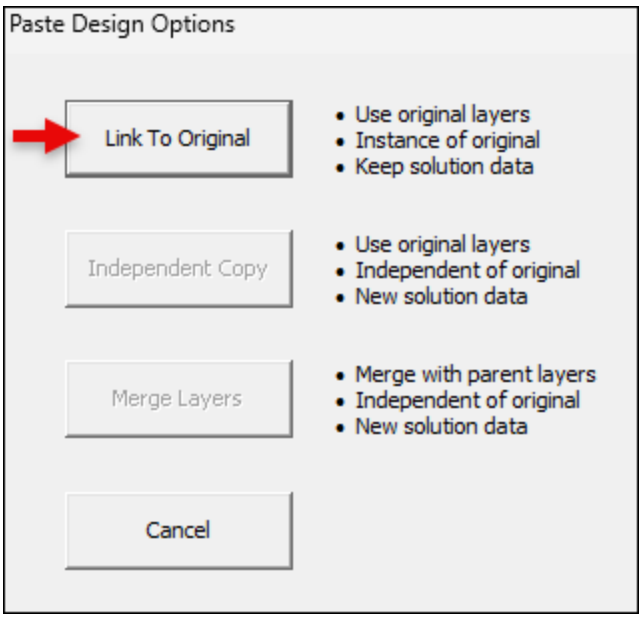
1. Begin with an HFSS 3D Layout design.
2. From **Tools** > **Options** > **General** > **Desktop Configuration**, click **Beta Options**. Under the HFSS 3D Layout subheading, ensure that **Q3D in General Mode** is enabled.
3. Create Circuit ports for all active components that will be connected to the layout in the Circuit schematic (e.g., voltage sources, transistors, etc.). Passive components (such as resistors, capacitors, and inductors) will automatically be included in the broadband parasitics model (Super SPICE model).
4. Add a Q3D solution setup. Ensure that **DC Resistance/Inductance** and **AC Resistance/Inductance** are enabled. Optionally, **Capacitance/Conductance** may be enabled.
5. Add a Frequency Sweep to the solution setup. Ensure that the type is **Interpolating**, and that the frequency points cover the entire range of interest, including DC point.

   > **Tip:**
   >
   > For higher accuracy of the power loss density distribution, consider applying additional mesh operations (Inside Selection > Length-Based).

6. Solve the Q3D solution setup in the HFSS 3D Layout design.
7. Insert a Circuit design in the same project.
8. From the **Project Manager**, click-and-drag the HFSS 3D Layout design over the Circuit design.

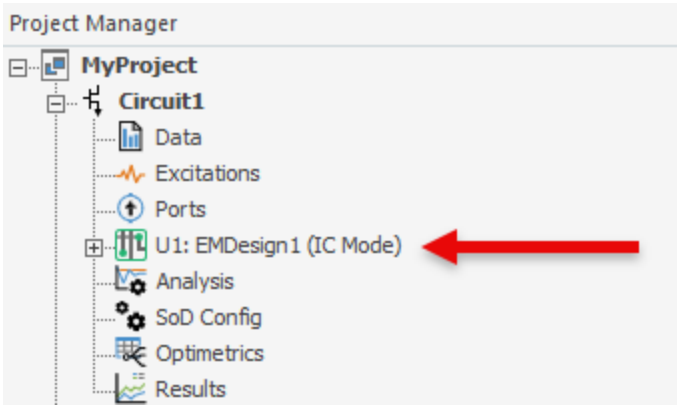
   The **Paste Design Options** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



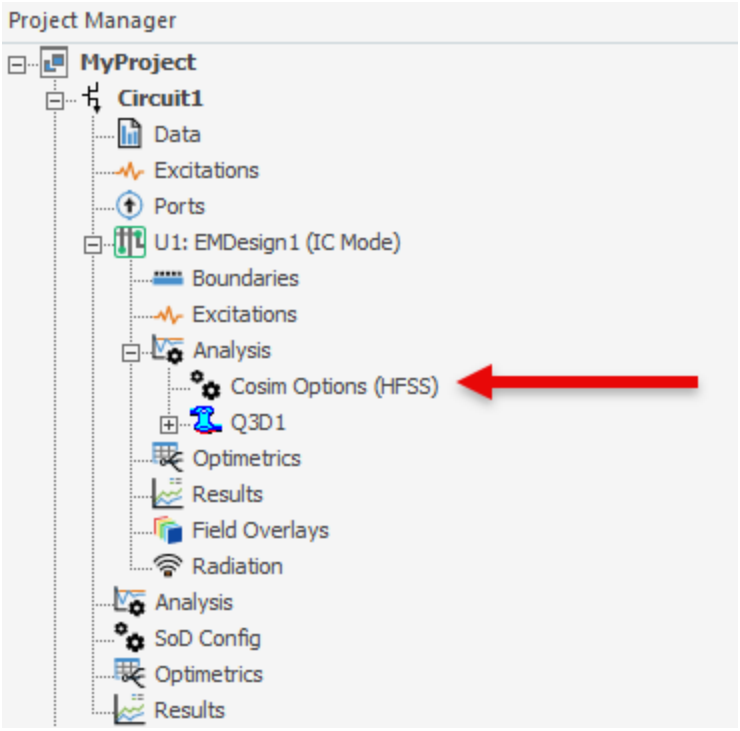
9. Click **Link to Original** to create a dynamic link between the designs.

   The linked HFSS 3D Layout design appears beneath the Circuit design in the Project Manager.

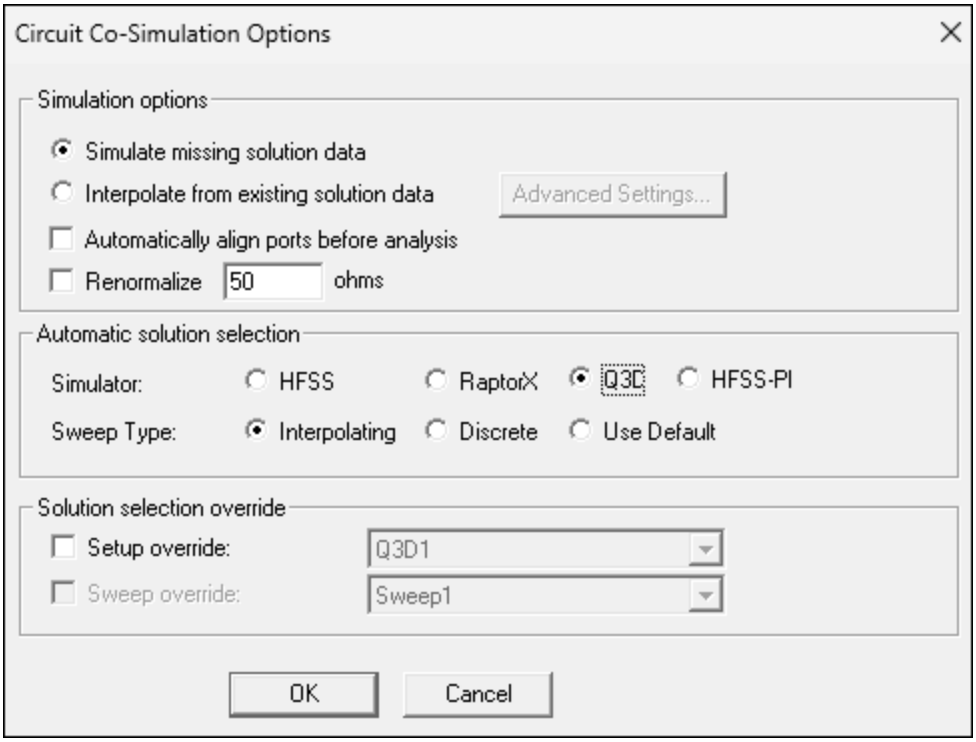


10. In the Project Manager, expand the HFSS 3D Layout design's branch. Under **Analysis**, double-click **Cosim Options (HFSS)**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Circuit Co-Simulation 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.

11. In the **Automatic Solution Selection** area, ensure the following are selected:
    - **Simulator**: Q3D
    - **Sweep Type**: Interpolating

    > **Note:**
    >
    > Alternatively, use the **Solution Selection Override** area to enable **Setup Override** and select the correct Q3D solution setup and sweep.

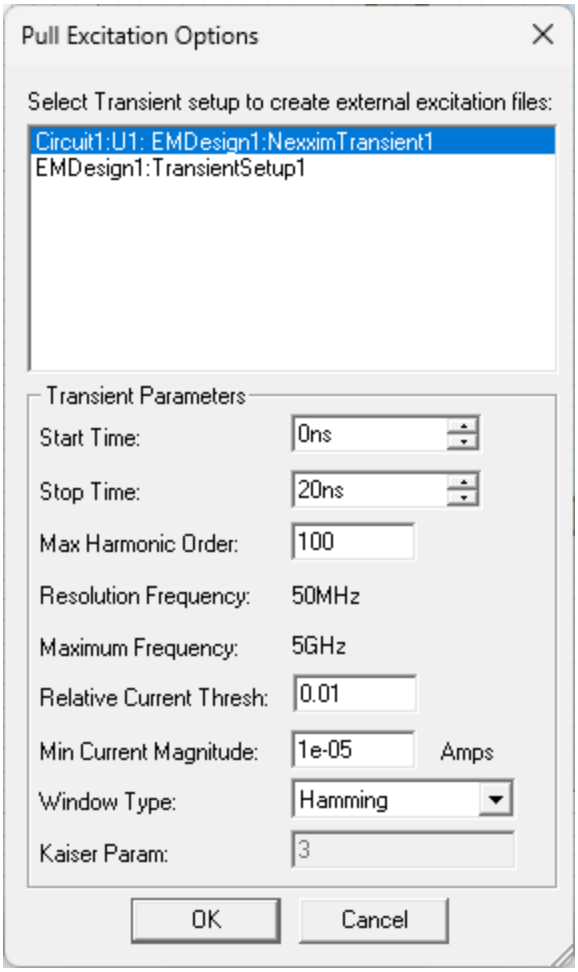
12. Under the Circuit design, add a Nexxim Transient Analysis solution setup. Analyze the setup.

    > **Note:**
    >
    > The Q3D component in Circuit will use the Super SPICE model rather than the State-Space model. Super SPICE is based on the Q3D RLGC component rather than the s-parameters used by the State-Space model.

13. In the HFSS 3D Layout design, add an SIwave DCIR solution setup.
14. From the Project Manager, right-click the SIwave DCIR solution setup and select **Pull Excitations**.

    The **Pull Excitation 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.



15. Select the Transient Nexxim setup and adjust the Transient Parameters. **Start Time** and **Stop Time** should be entered so that an integer number of periods (at least one full period) of the lowest harmonic are sampled after all currents in the Transient Circuit analysis have reached steady state.

    The currents at the ports are written to disk.

16. Run the SIwave DCIR simulation.
17. The HFSS 3D Layout design can be coupled to an Icepak design using **PCB Component** in Icepak.

Sanitize Layout

Sanitize Layout is an operation for cleaning power and ground nets in a project in order to fix certain alignment problems and complexities that may slow down simulation. It works by uniting the planes and traces for each net to be cleaned. Once the united planes are formed, portions which display trace like properties are converted into traces. Under certain conditions, these

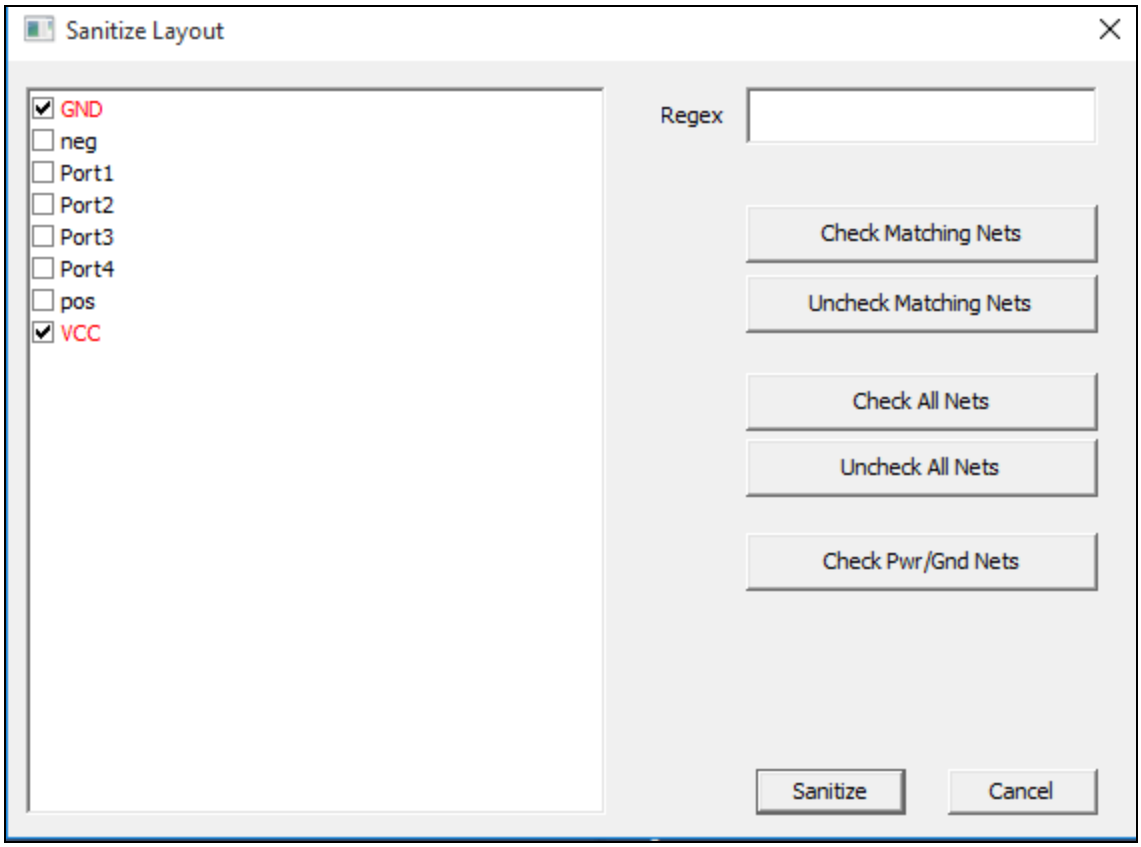
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

traces may be extended in order to improve the connection with vias, pads, and the remaining planes. Lattices which are originally formed of many traces become planes with cutouts after the sanitization process.

To sanitize the layout in the active project:

1. Click **Draw** > **Sanitize Layout**.

   The **Sanitize Layout** window opens, with all the power and ground nets selected.



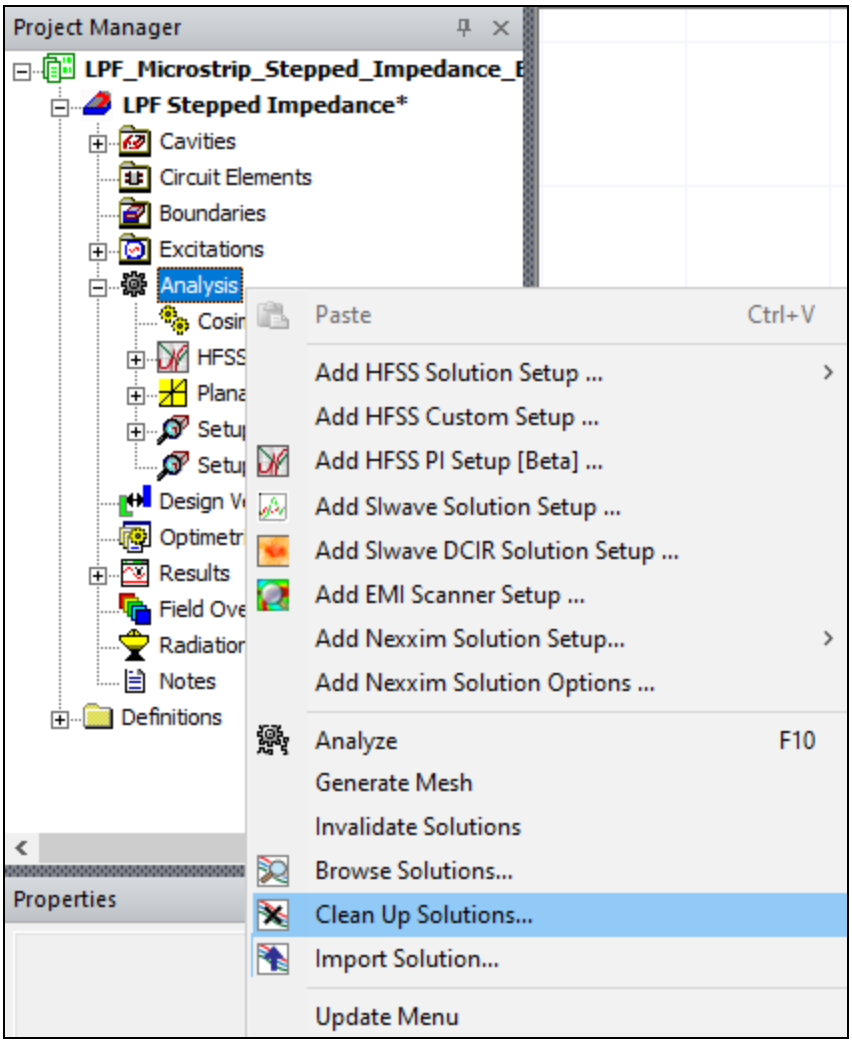
2. To use wildcards or regular expressions, enter the name of the nets you want to select in the **Reg Ex** field. Click **Check Matching Nets**.
3. To deselect matching nets selected using reg ex, click **Uncheck Matching Nets**.
4. Click **Check All Nets** to select all nets except dummy nets, or click **Uncheck All Nets** to deselect all nets.
5. Click **Check Pwr/Gnd Nets** to select all the power and ground nets.
6. Click **Sanitize** to run the sanitize layout on the selected nets.
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.

Clean Up Solutions

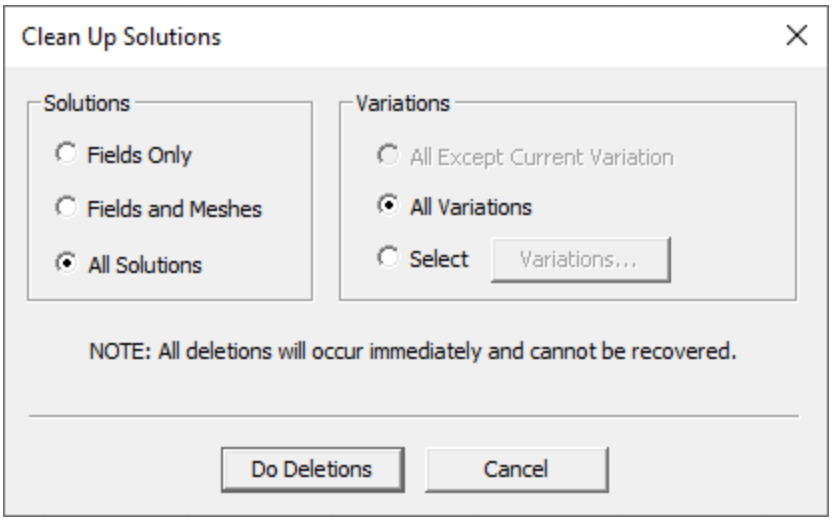
Follow these steps to clean up solutions.

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click the **Analysis** folder and select **Clean Up Solutions**



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Clean Up Solutions** window opens.



2. From the **Clean Up Solutions** window, choose which solutions to delete on the **Solutions** area (e.g., **Fields Only**, **Fields and Mashes**, or **All Solutions**).
3. Choose which variations to delete on the **Variations** area (e.g., **All Except Current Variation**, **All Variations**, or **Select Variations**).
4. To choose specific variations, choose **Select** and click **Variations** to open the **Select Variations** window. Then select variations to delete.
5. Click **Do Deletions** to delete all the selected meshes, matrices, field data, and variables. Any post-processing reports or field overlays that included deleted data are marked with an X in the project tree; they are invalid until new solution data is generated.

Adding a Technology to a Design (IC Mode Only)

Local design data and materials can be overridden with a predefined stackup (i.e., a technology file) without making permanent changes to the local parameters or materials. A technology file includes stackup layer material properties, units, thickness, elevation, and roughness data.

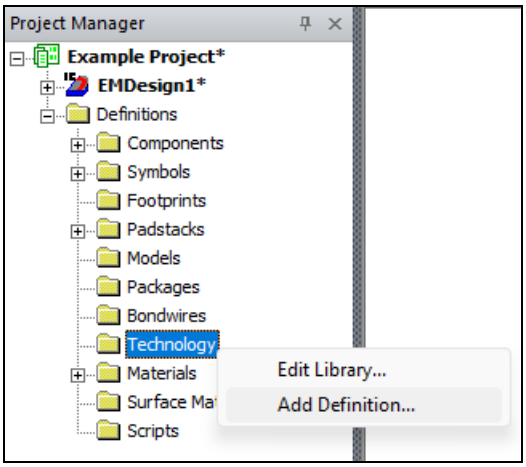
Follow these steps to add one or more technologies to a design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

If users import a GDSII file in IC mode, setup instructions differ, but an initial technology can be added to the design from the **GDSII Import** window. If a technology file was applied during setup, the technology's name should already appear in the **Layout** ribbon adjacent to the **Technology** drop-down menu. Additional technologies can be added later, as appropriate, according to the following steps. Refer to Importing GDSII Format Files in IC Mode.

1. From the **Project Manager** window, expand the **Project Tree** > **Definitions**. Then right-click **Technology** and select **Add Definition** to open the **Add Technology Definition** window.



2. From the **Add Technology Definition** window, navigate to and add the following files, as appropriate:

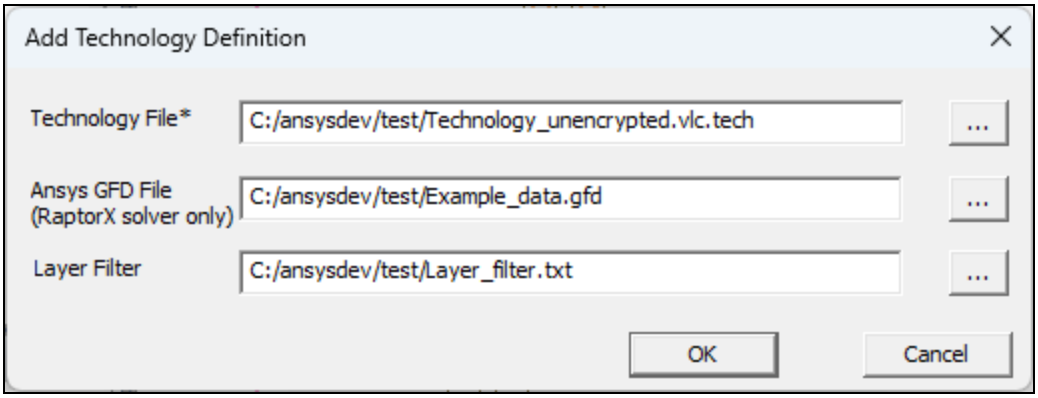
Note:

A **Technology File** is required. All selected files **must** reference compatible (i.e., in sync) data or errors will result.

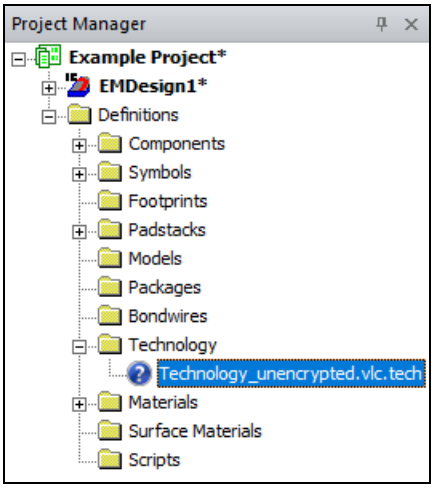
- **Technology File** — add a *.vlc.tech, *.ircx, *.itf, or .ict file.
- **Ansys GFD File (RaptorX solver only)** — import a Green's Function Data file (i.e., a *.gfd file). Refer to Viewing Green's Function Data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Layer Filter** — add a *.txt file with a list of signal/via layer names to specify their analysis. A technology file must either contain references to the same number of signal/via layers in a design's stackup or a layer filter can be used to specify which layers from the technology file to redefine within the stackup.

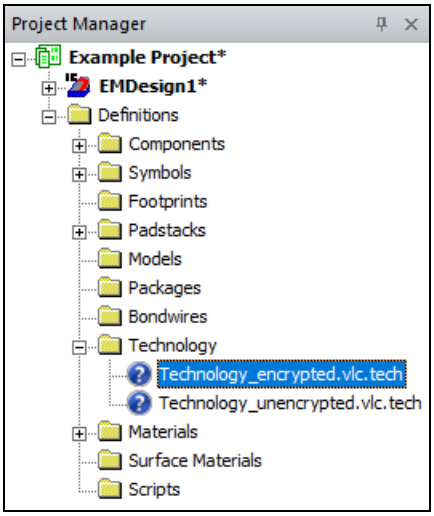


3. Click **OK** to add the new technology definition to the list of available technologies in the design.



4. Repeat steps **1-3** to add more technologies, as appropriate.

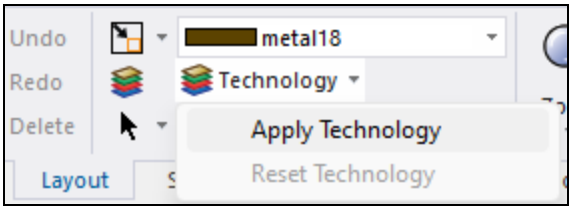
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

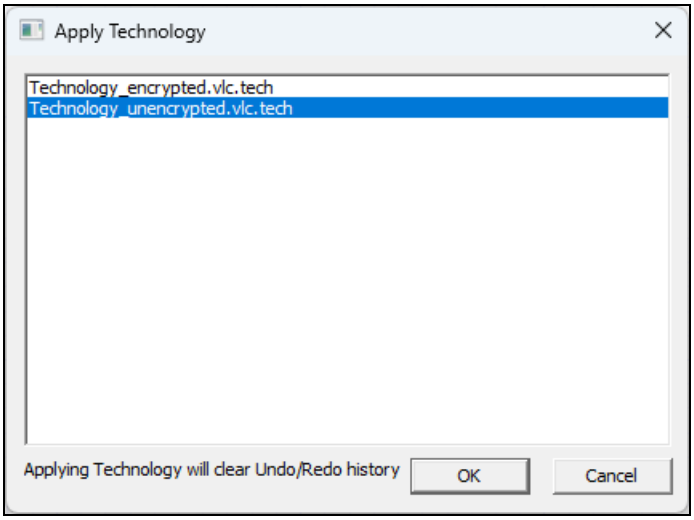
Adding an encrypted technology to the list of available technologies in the design will not immediately encrypt the design. Encryption is applied only after the technology is. Refer to step **6**.

5. Navigate to the **Layout** ribbon and select **Apply Technology** from the **Technology** drop-down menu.

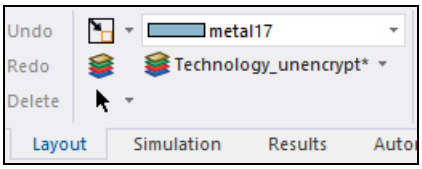


6. Once the **Apply Technology** window opens, select an appropriate technology from the list and click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



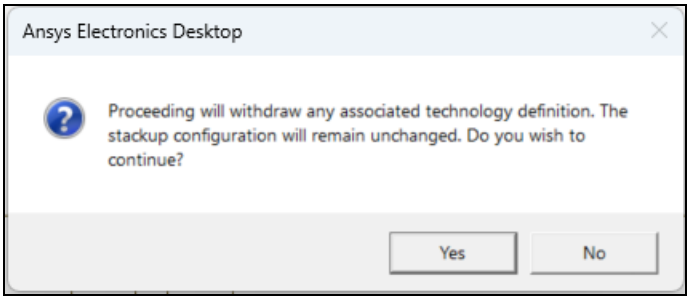
Once the technology is applied to the design, the technology's name appears adjacent to the **Technology** drop-down menu, the local stackup parameters are overridden, and the **Materials** list populates with the materials specified in the technology file.



7. To remove an applied technology and redefine a design with the local design data, navigate to the **Layout** ribbon and select **Reset Technology** from the **Technology** drop-down menu.

A message window appears. Users must confirm the reset by clicking **Yes**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

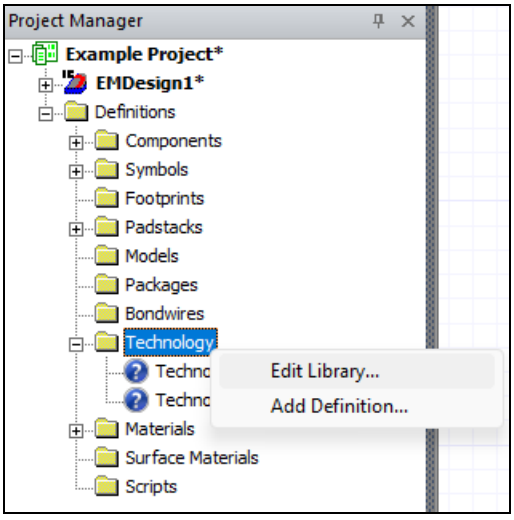


Note: Reset must also be performed to modify a technology that has been applied to a design. To make any appropriate changes to a technology, it must first be disassociated from the design (i.e., **Reset**), if applicable, then removed from the **Technology Library**. Finally, after an appropriate changes have been made to the technology file and optional **Green's Function Data** and **Layer Filter** files, the technology must be re-added to the **Technology Library** Refer to steps **1**-**3** in this section and steps **1**-**2** in Viewing the Technology Library.

Viewing the Technology Library

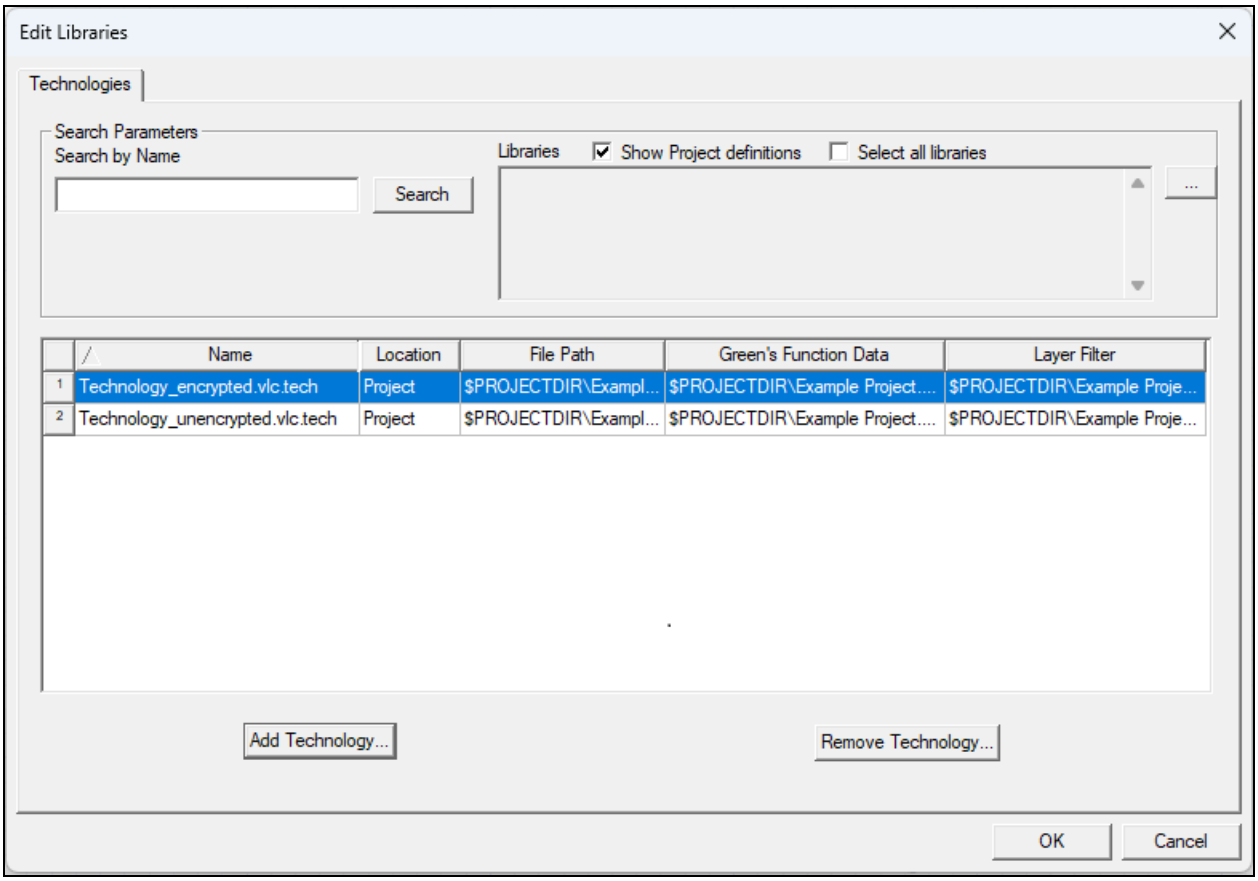
Follow these steps to view, add, and/or remove technologies from the **Technology Library**.

1. From the **Project Manager** window, expand the **Project Tree** > **Definitions**. Then right-click **Technology** and select **Edit Library** to open the **Edit Library** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

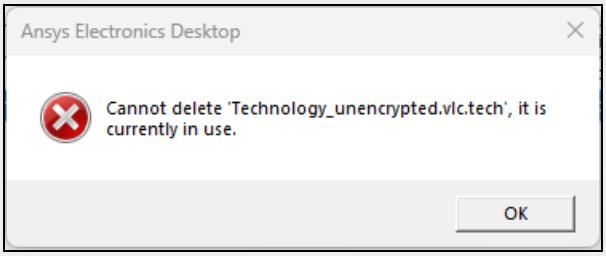
2. From the **Edit Library** window, users can quickly view where technology assets are stored, whether **Green's Function Data** and/or a **Layer Filter** are attached to a file, search the contents of technology files added to the design, (i.e., **Search Parameters**), add more technology definitions (i.e., **Add Technology**), and remove files from the design that are no longer appropriate (i.e., **Remove Technology**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

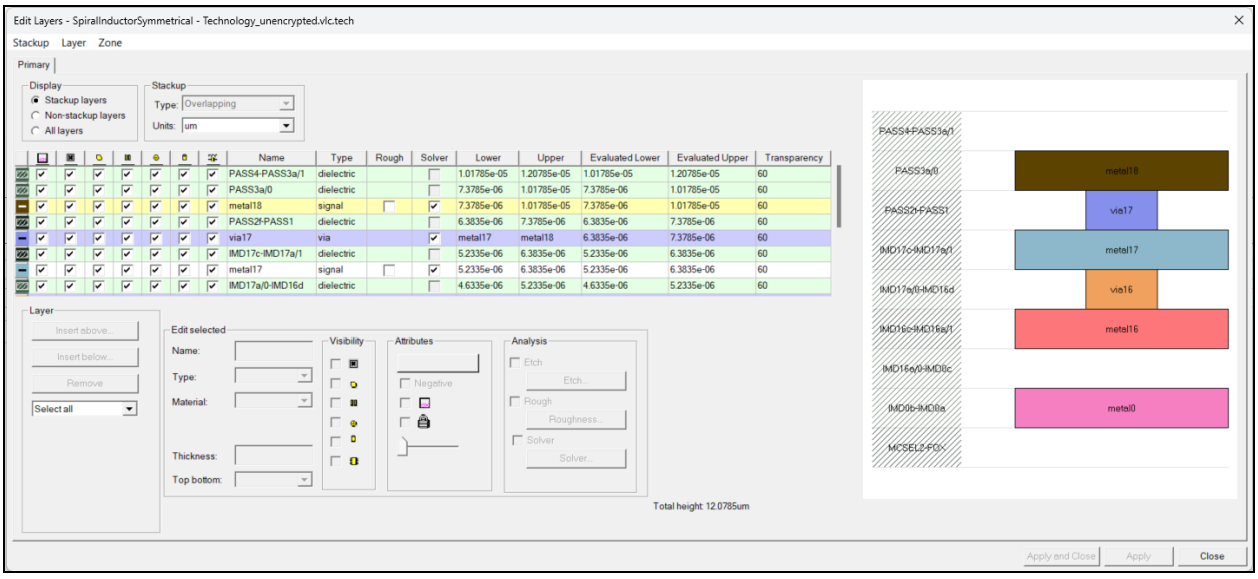
Note:

A technology currently applied to the design cannot be removed. An error message appears if users attempt to do so.



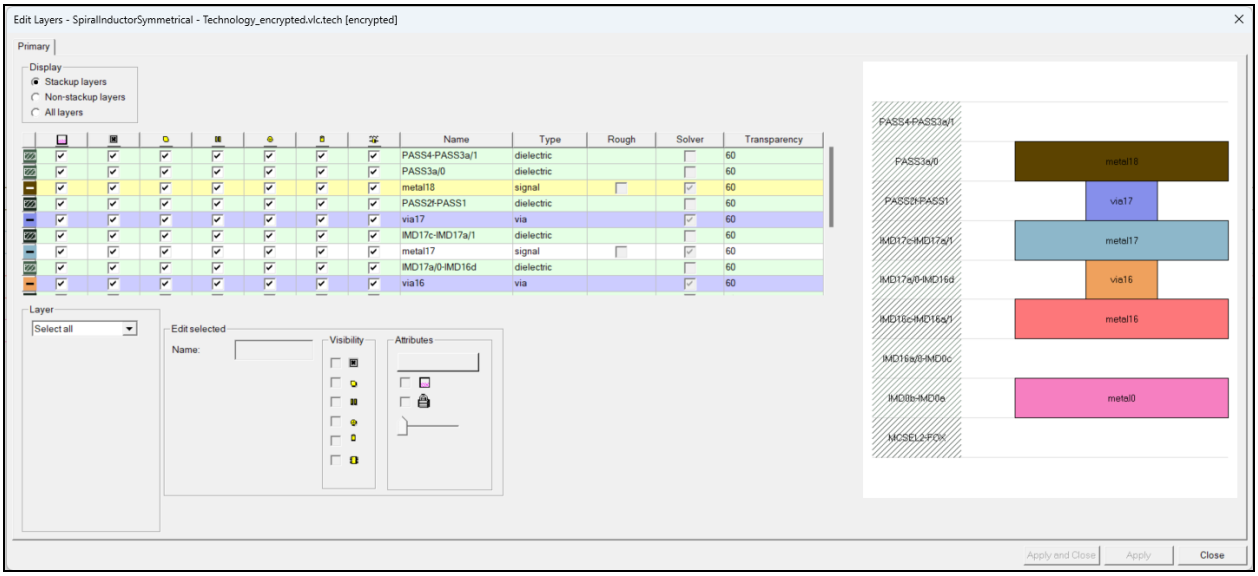
Applying Encrypted Technology Files

Encrypted technologies define the design with obfuscated parameters and materials, although the design can still be analyzed. Among the more obvious examples of an encrypted technology are the visualization of the design in the **Layout Editor** as well as the stackup parameters and representations in the **Edit Layers** window. The **Edit Layers** window typically has columns identifying layer material, thickness, and elevation (as shown in the following example):

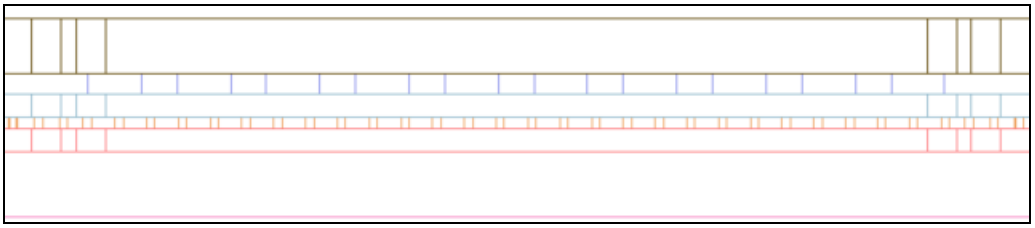


Once a design is encrypted, the columns identifying materials and physical parameters of the layers (e.g., material, dimensions, et cetera) are hidden.

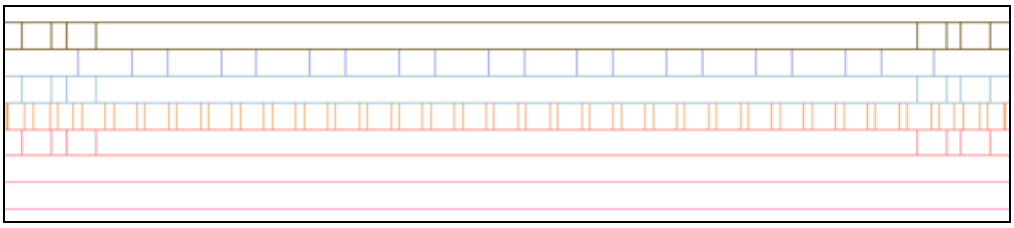
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Similarly, before applying an encrypted technology, the **Layout Editor** represents the design accurately, displaying all geometry according to its material type, position, dimensions, and layer thickness. All parameters are edible.



After encryption, geometry and port location(s) remain visible, but layers in the **Layout Editor** are normalized (e.g., replaced with equidistant layers) creating a noticeably more symmetrical design and less accurate representation. Parameters are largely hidden and uneditable.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Once an encrypted technology is applied to a design, the design is locked and encryption cannot be removed from the component until the technology is replaced by an unencrypted technology with the same signal/via layer configuration.

Note:

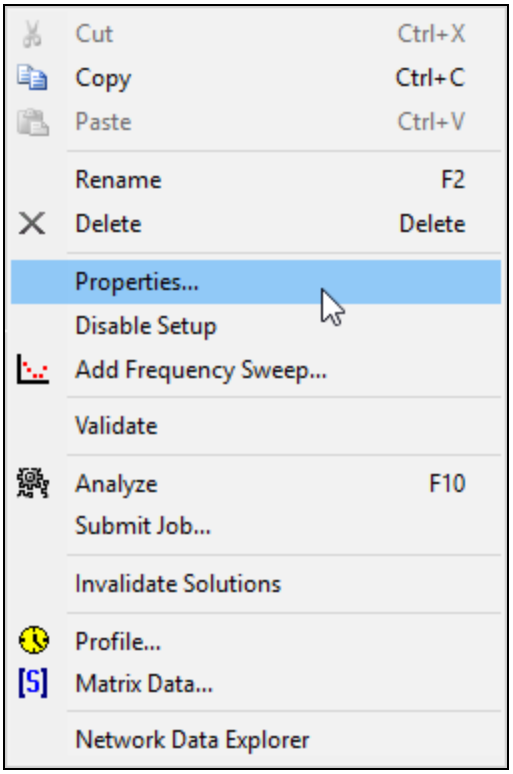
Encrypted technologies are only supported by HFSS, Q3D, and RaptorX simulations.

Modifying a Solution Setup in HFSS 3D Layout

After adding a solution setup, change the solution's setup parameters.

To reopen the setup window, do one of the following:

- From the Project Tree, double-click the solution setup.
- From the Project Tree, right-click the solution setup and select **Properties**.



After right-clicking, **Rename**, **Delete**, and **Disable** options are also available.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Disabling an Analysis Setup

You can activate or deactivate a setup definition. Disabled setups have a shaded setup icon, and do not allow analysis of any sweeps until enabled. To deactivate 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 setup definition and select **Disable Setup**. (To reactivate the sweep, right-click and select **Enable Setup**.)

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. HFSS 3D Layout 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:

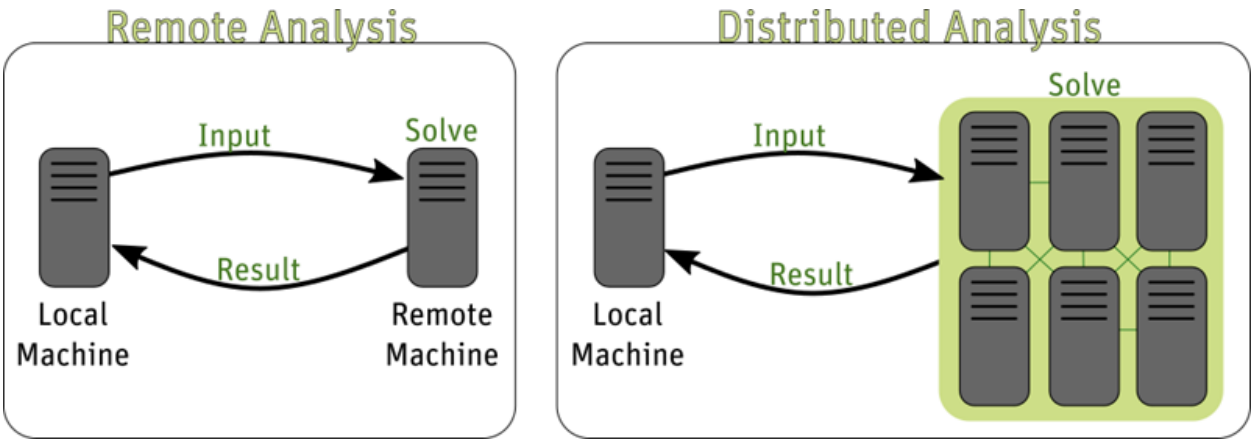
- Remote Analysis
- Distributed Analysis
- Ansys Cloud Burst Computing Support for HPC
- Large Scale DSO for Parametric Analysis
- Interactive Scheduler Jobs
- HPC Integration
- Multi-Step Job Submission
- Distribution Command Line Options
- GPU Acceleration

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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This can also be extended into distributed analysis, where a specified analysis, if supported, is concurrently solved on multiple machines.



> **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Ansys Electronics Desktop must be accessible from all remote machines as well as on the local machine. If the analysis uses MPI, then the path of the Ansys Electronics Desktop installation must be the same on all of the machines used for the analysis (remote and local). This may be a shared network path accessible from all hosts. Alternatively, it may 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.

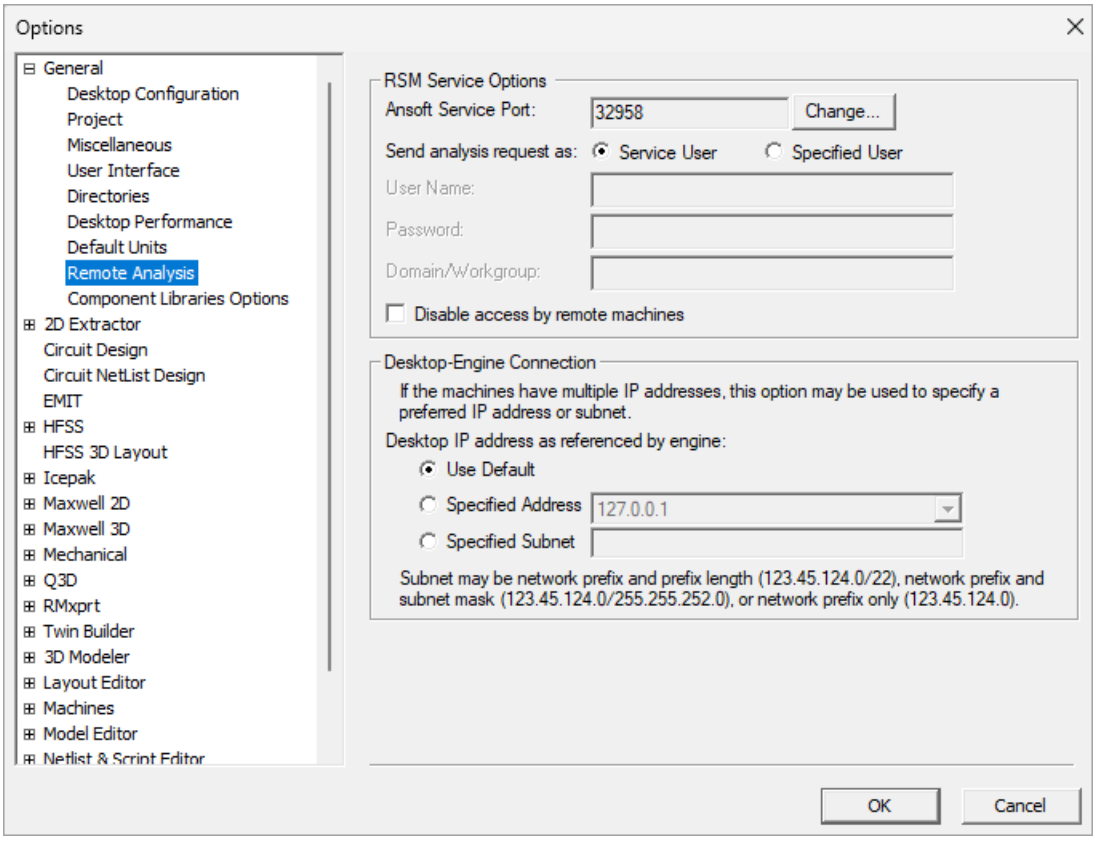
Configuring the Local Machine to Solve Remotely

To set HFSS 3D Layout 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.

Troubleshooting

> **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.

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.

HFSS 3D Layout 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.

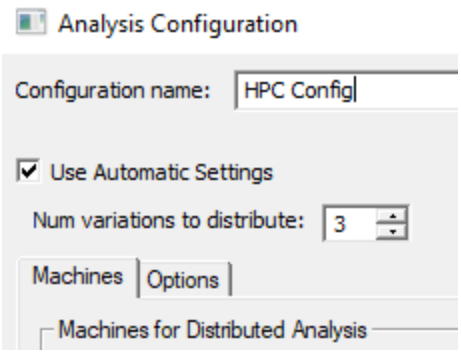
> **Note:**
>
> Communication between machines in remote analysis and distributed analysis can drastically affect performance. Use of a high-speed network system, like Gigabit or Infiniband, is recommended for optimal performance.

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 enabled on the active Analysis Configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When running a suitable design with this feature enabled, the Solution Profile will show multiple frequencies being adapted at the same time.

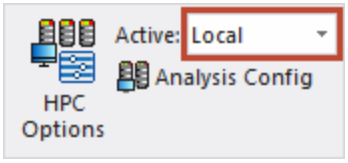
Configure a Distributed Analysis

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.



Distributed Analysis Configuration

After you have selected the active distributed machine configuration, you can edit its configuration in the **Analysis Configuration** window.

To edit the configuration:

1. Access the **Analysis Configuration** window one of three ways:
    - From the **Simulation** tab, click **Analysis Config**.
    - Click **Tools** > **Edit Active Analysis Configuration**.

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 **HPC and Analysis Options** window (opened from the **HPC Options** icon on the **Simulation** ribbon or **Tools** > **Options** > **HPC and Analysis Options**), select the configuration and click **Edit**.

> **Note:**
>
> You can also access the Analysis Configuration window from the **HPC and Analysis Options** window by clicking **Add** or **Copy**. When using **Add** or **Copy**, the steps are the same as below, but you will need to specify a **Configuration Name**. The name cannot be empty and it cannot be a reserved name.

The **Analysis Configuration** window appears.

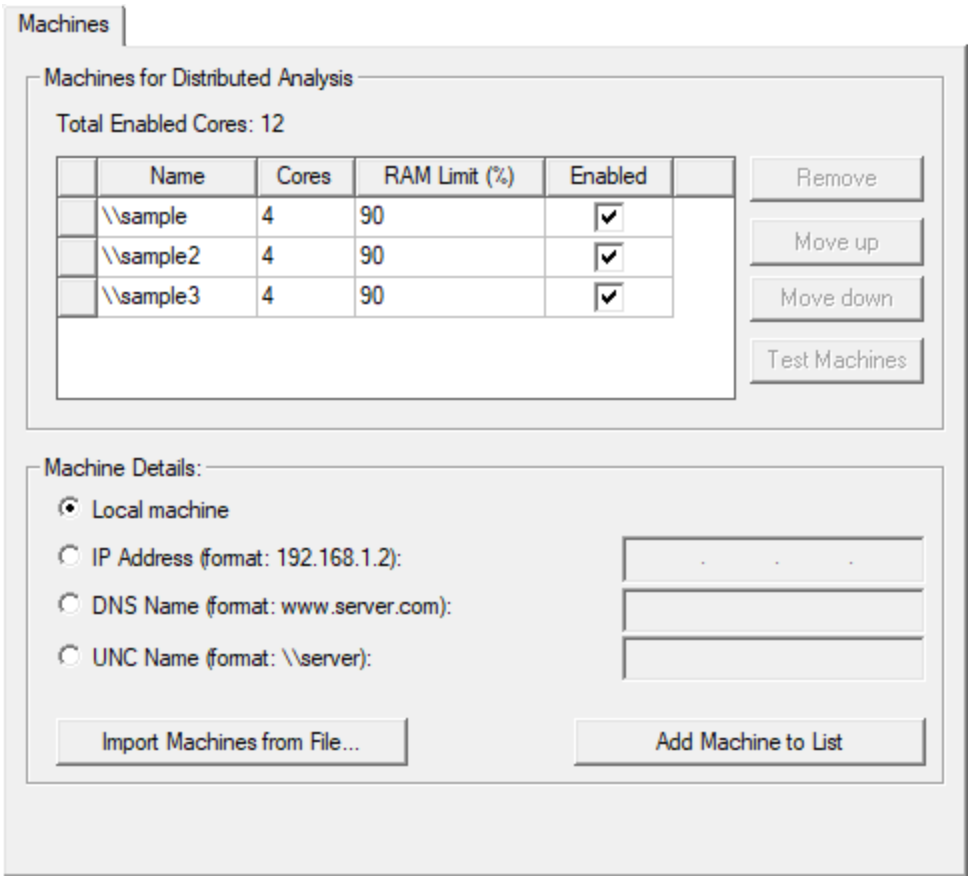
> **Note:**
>
> Available options may vary slightly depending on the solver and design.

2. If automatic settings are supported, the **Use Automatic Settings** check box allows you to select this option and set parameters based on the best use of available resources for the current analysis. Deselecting it enables the **Job Distribution** tab, where you can assign resources manually.
3. Use the tabs to navigate through the rest of the settings:
   - **Machines** – contains the machine list. You can add, enable, remove, test, and reorder machines from the list.
   - **Job Distribution** – when **Use Automatic Settings** is deselected, this tab appears and allows you to select job distribution settings manually.
   - **Options** – allows you to specify additional options, depending on the design type.
4. Click **OK** to save the configuration settings.

Distributed Analysis Configuration - Machines Tab

On the **Analysis Configuration** window, the **Machines** tab allows you to provide machine information, either by specifying remote machine details, or by importing a list of machines from a file. You can then add, enable, remove, test, and reorder machines from 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.



Manually Adding a Machine

You can add the **Local machine**, or add a machine by **IP Address**, **DNS Name**, or **UNC Name**. Select the applicable radio button, enter the machine's information in the specified format, then click **Add Machine to List**.

> **Important:**
>
> The remote machines must have the same Ansys Electromagnetics Suite version installed on the same OS version, and have active RSM service.

The added machine(s) will appear under **Machines for Distributed Analysis**.

Importing a Machine List

You can import a machine list from a text file. Each line of the file should contain a machine's information in the following format:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<MachineName>:<NumTasks>:<NumCores>
```

Machine List Details

The **Machines for Distributed Analysis** area lists machines in 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** and **Move down** buttons. To remove one or more machines, select the machine(s) and click **Remove**.

The following columns can appear in the list:

- **Name** – the machine name.
- **Tasks** – this column allows you to specify the number of tasks a given machine will perform simultaneously. *Each separate solver or instance counts as one task.*

  If you have selected **Use Automatic Settings**, this column does not appear because there is no need for it.

- **Cores** – specifies the total number of cores that will be used on the given machine. The total number of Tasks and Cores are described just above the machine list.

  For distributed tasks, the software will allocate the total cores on a given machine to that machine's tasks. If a machine with 8 cores is running 2 distributed tasks, the software will automatically allocate 4 cores to each task. If it is running 4 distributed tasks, each gets 2 cores. And if it is running 3 distributed tasks, the first two tasks get 3 cores and the last task gets 2 cores. *The number of Cores must always be greater than or equal to the number of Tasks.*

  For a given variation (for example, frequency or geometry), you should 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 example, with two machines, one with eight cores and another with four, assuming that the memory is proportionally equivalent, you could assign two tasks for machine 1, and one task for machine 2, giving all tasks the same number of cores.

- **RAM Limit (%)** – specifies the maximum percentage of each machine's RAM you would like to be in use by the solver.
- **Enabled** – use the check boxes to enable or disable machines.

In general, Ansys Electromagnetics Suite solvers use machines in the distributed analysis machines list in the order in which they appear. If you select a distributed configuration (rather than Local) from the Toolbar menu and you launch multiple analyses from the same UI, Ansys Electromagnetics Suite solvers select 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, the solver chooses the first machine in the list. If another simulation is launched while the previous one is running, and this

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulation requires two machines, the solver chooses machines 2 and 3 from the list. If the first simulation then terminates and we launch another simulation requiring three machines, the solver chooses 1, 4, and 2 (in that order).

Testing 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 that no other Ansys EM processes are running on the machine. To do this, select one or more machines and click **Test Machines**.

The **Test Machines** dialog box appears. The test goes through the current machine list and gives a report on the status of each machine. A progress bar shows testing progress. When the test is complete, click **OK** to close the dialog box. If you need to disable or enable machines based on the report, you can do so in the **Analysis Configuration** window.

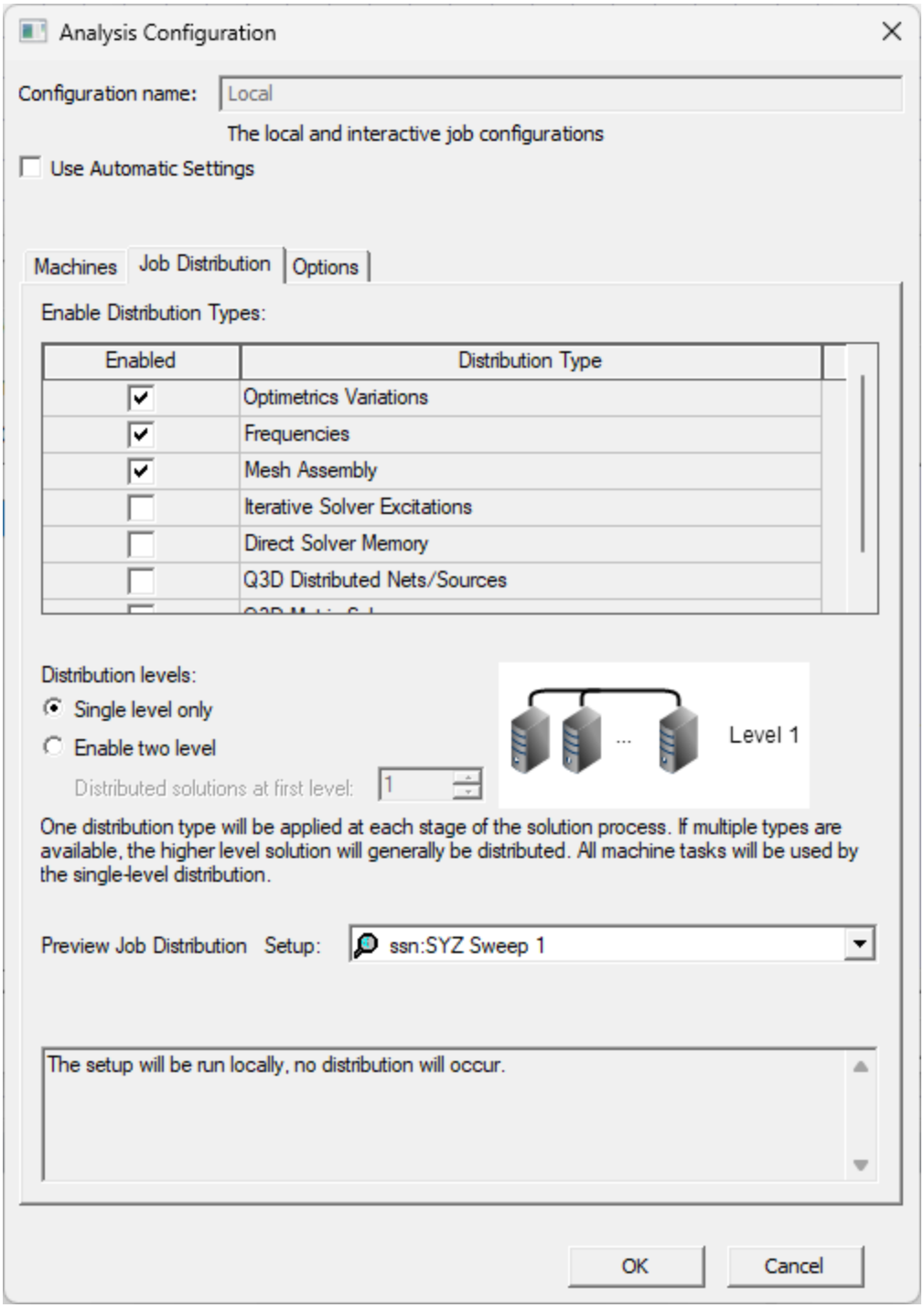
Distributed Analysis Configuration - Job Distribution Tab

On the **Analysis Configuration** window, the **Job Distribution** tab allows you to manually enable specific types of job distribution and enable multi-level solves.

The **Job Distribution** tab is disabled if you selected **Use Automatic Settings**.

> **Note:**
>
> Different design types have different job distribution types. Two of the options shown in the distribution list are beta features that improve performance on large designs by parallelizing either nets/sources or the matrix itself across MPI processes. Beta features must be activated prior to use. Refer to Activating the Q3D Distribution List Options Beta Feature for instructions.

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 distribution list allows you to specify which job distribution types to allow for the current analysis configuration. Use the check boxes enable and disable distribution types. At solve time, HFSS 3D Layout automatically selects the best distribution type from the enabled types.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enabling a distribution type does not mean it will be used. It must be also allowed by the solve setup. If you enable a distribution type for a given setup, and distribution is allowed, the preview window updates to describe the distributions. Note that enabled distribution types apply to all setups of the given design type, so it is possible for different setups in a design to be solved using different distribution types.

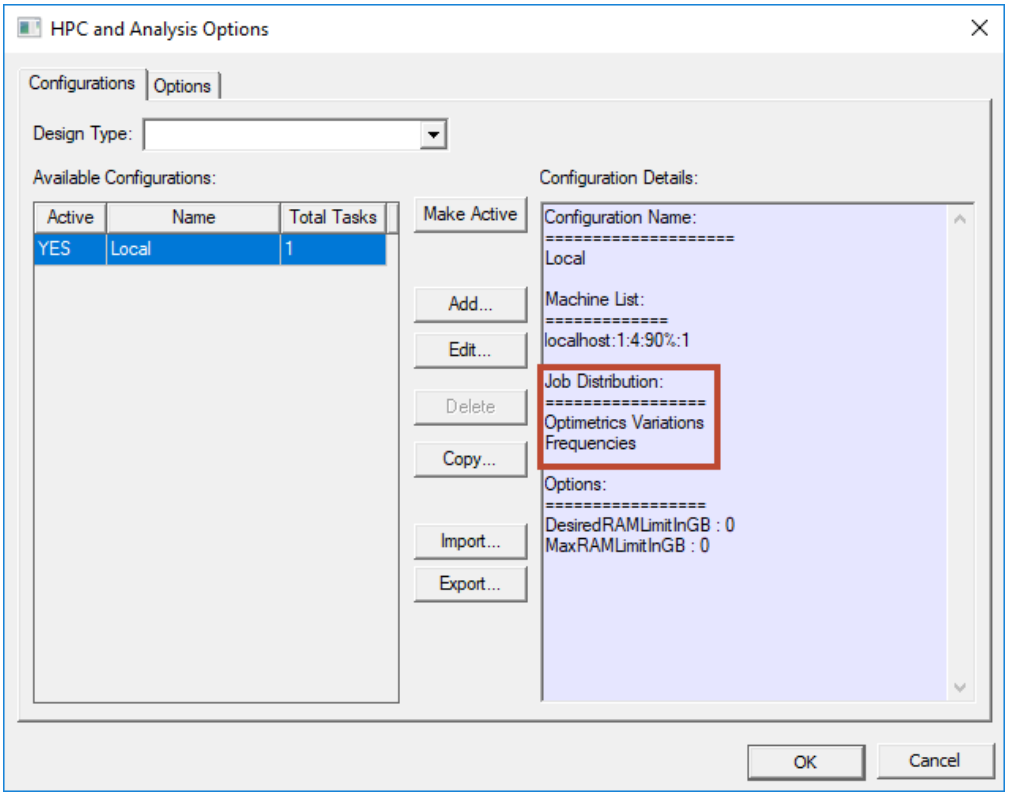
The concurrent initial mesh generation workflow with Distributed Mesh Assembly relies on the MPI based distribution technology inside the MeshAssemblyManager. The decision whether to launch sequential or parallel mesh generation is based on the combination of the number of individual meshes present, the HPC setting, and the number of tasks available.

- If there's only one single mesh or if curvilinear contact exists: then normal initial mesh workflow with single G3dmesher will be launched.
- If there are more than one individual mesh and no curvilinear contact exists:
    - If Use Automatic Setting is not selected, under the **Job Distribution** tab, if "Mesh Assembly" is not enabled, then meshes will be generated sequentially. If "Mesh Assembly" is enabled and user assigns more than 1 tasks, then parallel mesh generation will be launched.
    - If Use Automatic Setting is selected and user assigns more than 1 core, then parallel mesh generation will be launched.
    - If Use Automatic Setting, the number of MPI tasks will be the number of cores user assigned or the number of geometries to be meshed, whichever is smaller. If there are more geometries than cores, the geometries will be dynamically assigned to tasks. If there are more cores than geometries, the remaining available cores will provide some of the tasks with multi-thread capabilities.

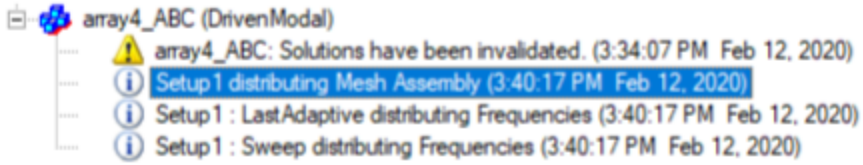
All the detailed progress information from the Mesher is suppressed and the progress will report the number of meshes being finished. All the mesh profiles will be available under profile report and the mesh feedback will also be available under mesh feedback tool.

Enabled distribution types are listed when you select a configuration in the **HPC and Analysis 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.

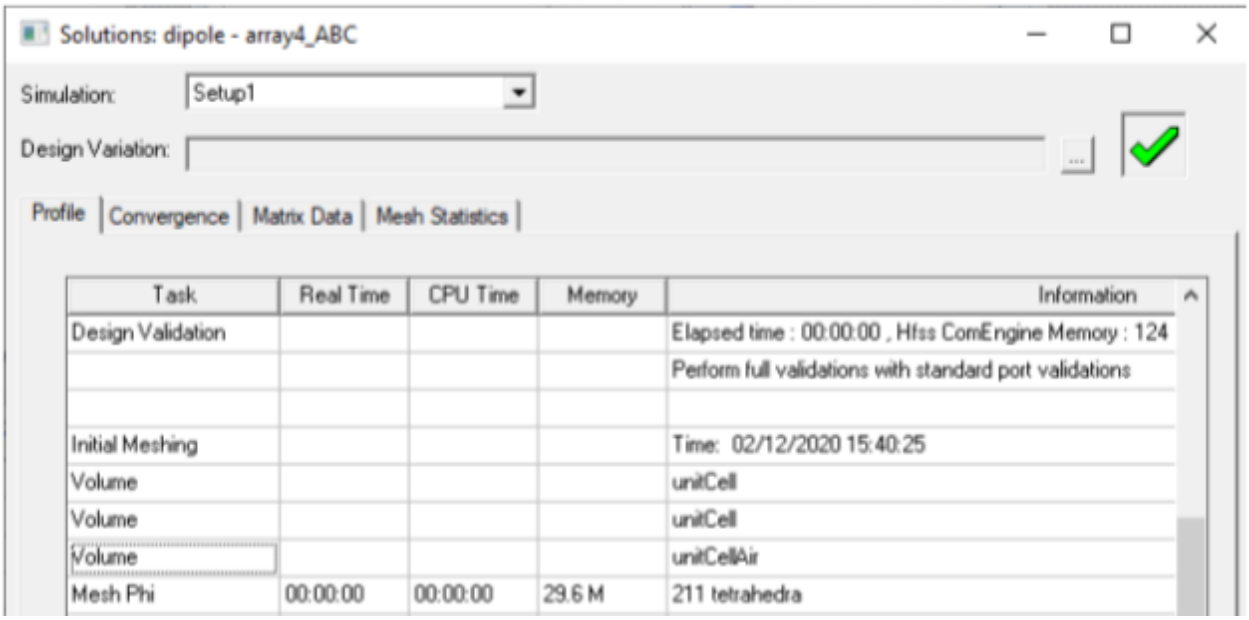


When you run a simulation, the **Messages** window describes distributions.



When you view the Solution Profile, distributions for unitCell show as parallel Volume Tasks.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



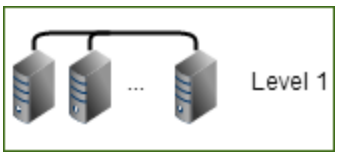
Distribution Levels

For products and designs that support two-level distribution, you can select either single or two-level distribution.

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.

Single-level Distributions

In a single-level distribution, one distribution type is applied at each stage of the solution process. Common stages include LastAdaptive, Sweep, and Parametric. All machine tasks will be used by the single-level distribution.



Supported distribution types include:

- Optimetrics Variations
- 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.

- Iterative Solver Excitations(for Technical details, see Multiprocessing and the Iterative Solver)
- Direct Solver Memory(for Technical details, see Distributed Memory Solutions with HFSS)
- Q3D Distributed Nets/Sources
- Q3D Matrix Solver

Solver distributions require MPI. See: Setting HPC and Analysis Options.

Parallel distribution types (such as Optimetrics Variations, Frequencies, and Excitations) do not require distribution. If these types are not able to distribute, the simulation can run sequentially.

Memory distribution types (such as Direct Solver Memory and Domain Solver) are set to require distribution. If these types are enabled, the software will assume that distribution is necessary to extend the simulation scale or add fundamental solution capabilities.

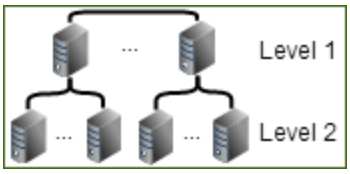
When multiple distribution types are available, the higher-level solution will generally be distributed. For example, when both Optimetrics Variations and Iterative Solver Excitations are enabled, Optimetrics Variations will be distributed. When both Optimetrics and Mesh Assembly, are selected Optimetrics are distributed. Domain Solver, Direct Solver Memory, Q3D Matrix Solver, and Q3D Distributed Net/Sources are exceptions because they are required; even though they are lower level, these types are distributed instead of parallel distribution types.

Two-Level Distributions

Selecting **Enable two-level** enables the **Distributed solutions at first level** box.

In a two-level distribution, the first level distributes the specified number of solutions. Each solution will then 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.

For two-level distribution, the total number of tasks must be greater than or equal to the number of tasks for level 1.



The following are examples of two-level distributions:

- A parametric setup distributing Optimetrics variations as level 1 and iterative solver excitations as level 2.
- A parametric setup for a non-transient problem distributing optimetrics variations as level one and frequencies as level 2.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A frequency sweep distributing frequencies as level 1 and direct solver memory as level 2.
- A parametric setup for a transient network problem distributing optimetrics variations as level one and transient excitations as level 2.

The **Analysis Configuration** window displays a preview of the job distribution, if allowed for that configuration.

For more information, see: Two-Level Distribution Guidelines.

Activating the Q3D Distribution List Options Beta Feature

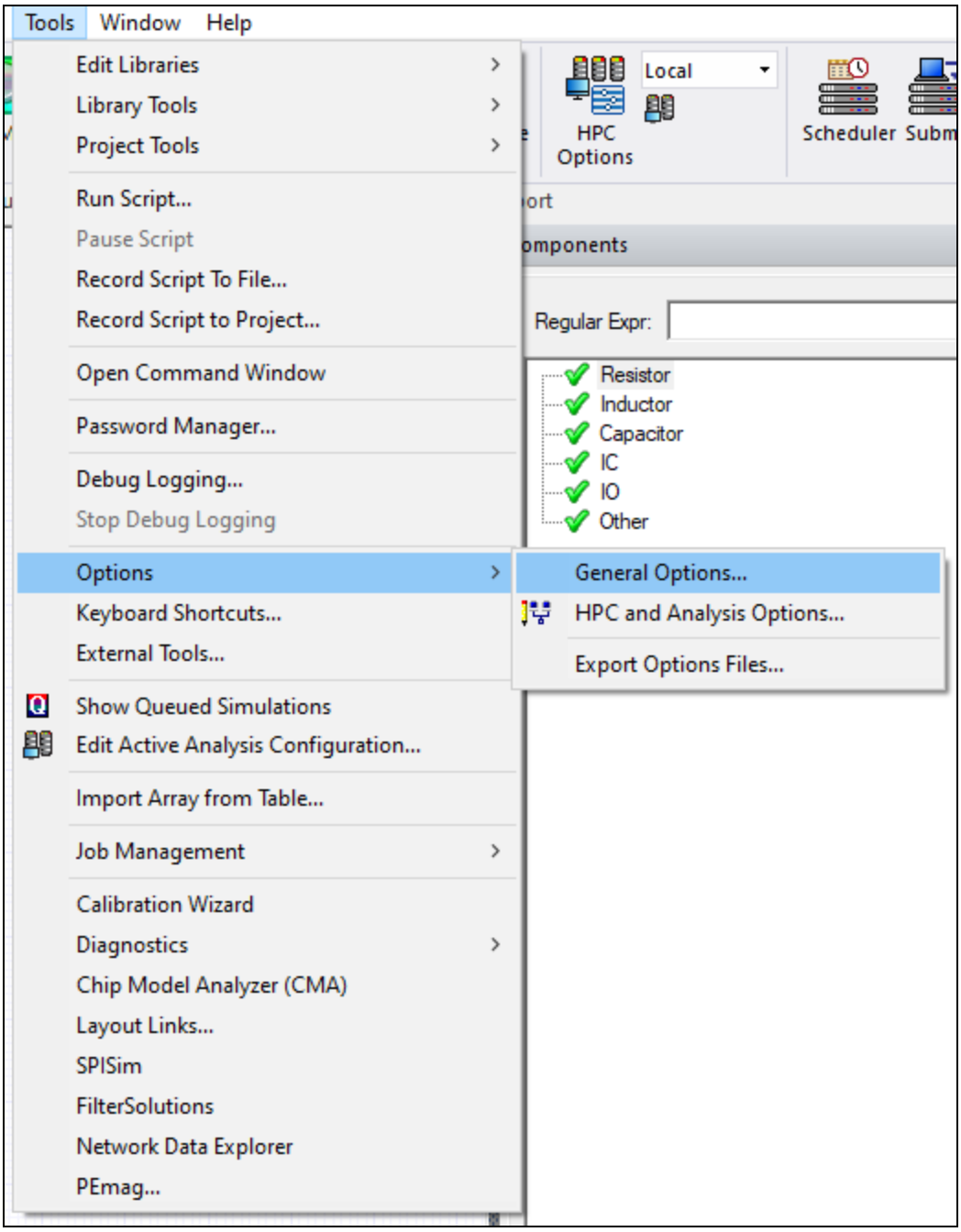
Each of the new Q3D job distribution types improve performance on large designs by parallelizing different parts of the solution. **Distributed Nets/Sources** targets problems with hundreds of nets or sources by splitting MPI processes into groups. Each group holds the full matrix, while different nets are distributed across the groups. This improves scaling when solve time is significantly greater than setup time. The solver automatically chooses the optimal number of groups based on memory profiling and the solve/setup ratio. **Matrix Solver** partitions the matrix itself across MPI processes so that very large problems can be solved when a single machine's memory is insufficient. By redistributing the mesh so that each MPI task handles only a partial mesh, and by parallelizing tasks, these new distribution types allows the simulation to handle designs containing 10 million+ mesh elements.

> **Important:**
>
> Distributed Memory AC RL solutions are recommended for large designs that cannot fit on a single node. Using too many tasks for smaller meshes may degrade performance. For optimal results, use a smaller number of cores per task (ideally 4).

Follow these steps to add **Q3D Distributed Nets/Sources** and **Q3D Matrix Solver** to the distribution list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

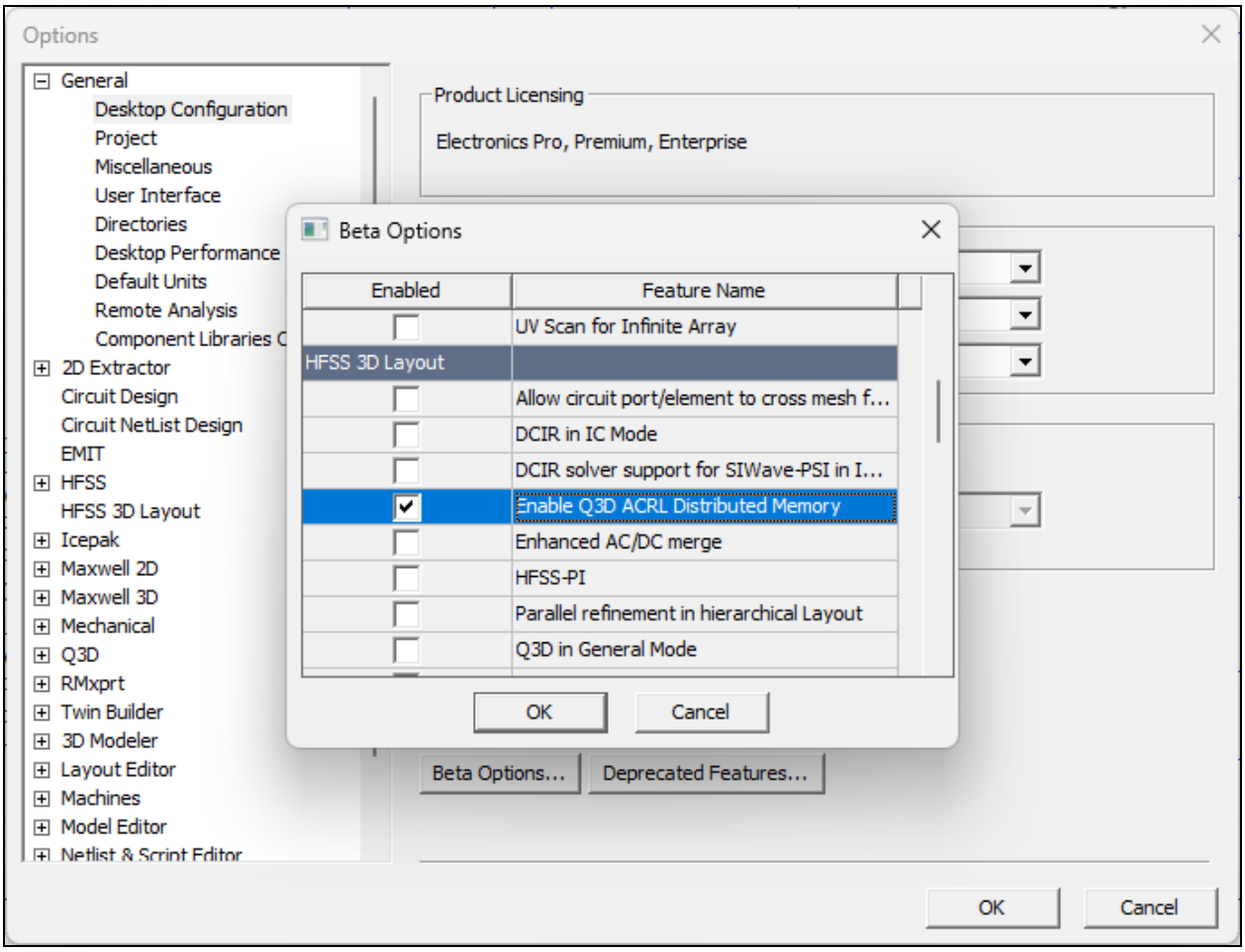
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window



2. Expand **General** > **Desktop Configuration** and click **Beta Options** to open the **Beta Options** window

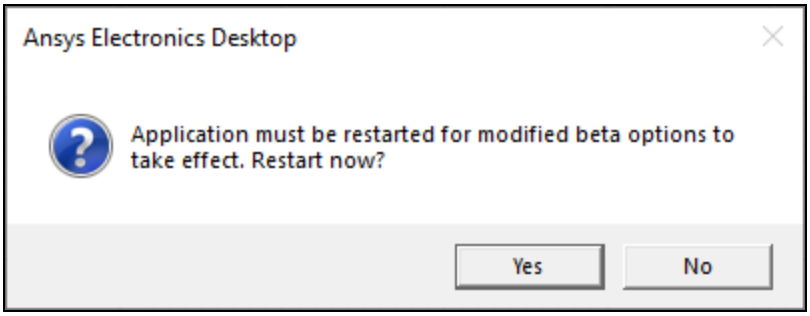
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scroll down to the **HFSS 3D Layout** section and check the box adjacent to **Enable Q3D ACRL Distributed Memory**.



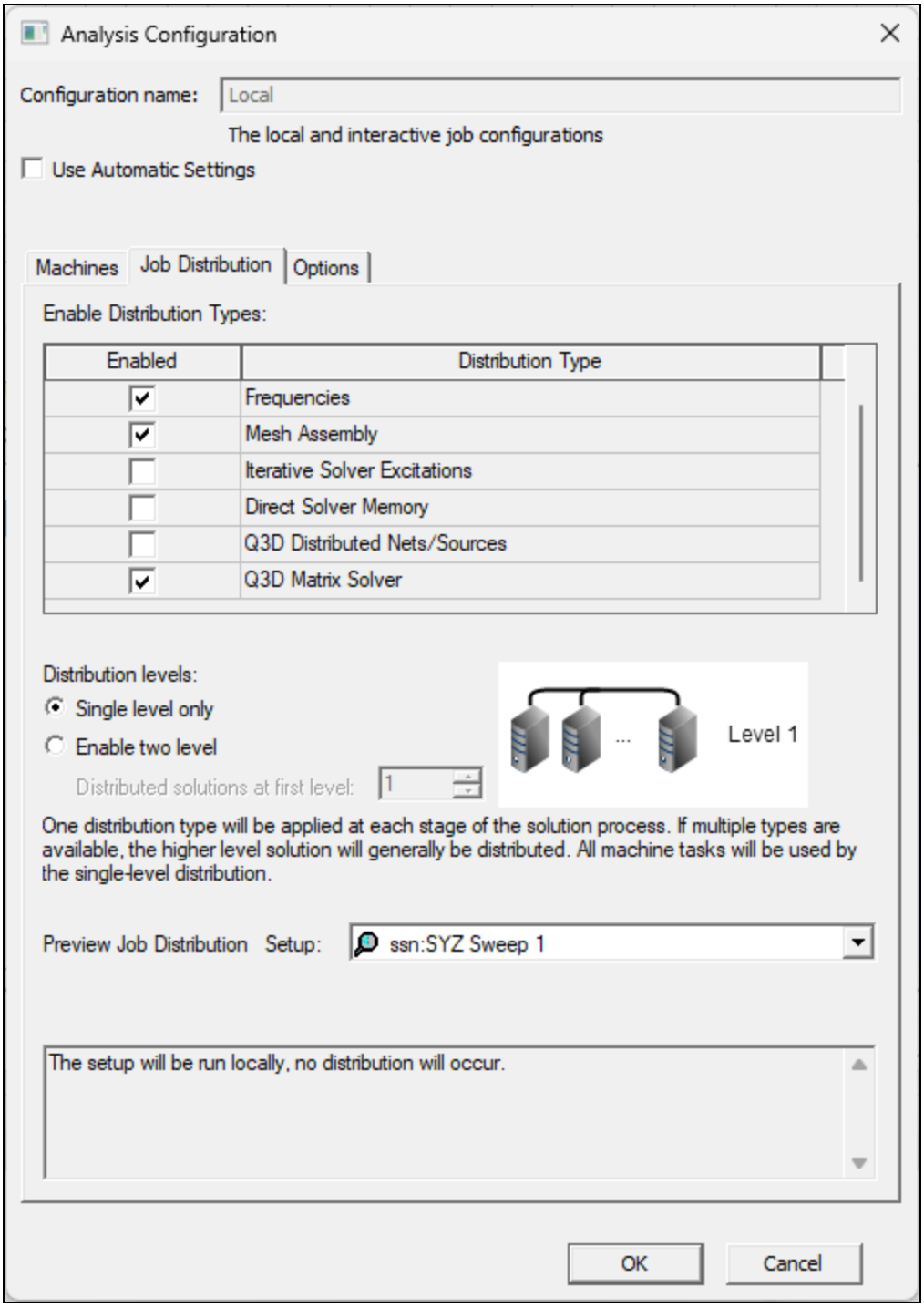
4. Click **OK** to close the **Beta Options** window.
5. Click **OK** to close the **Options** window and open a prompt to restart the software. Users may be asked to save their work.

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 **Yes** to restart immediately.
7. After restarting the software, **Q3D Distributed Nets/Sources** and **Q3D Matrix Solver** will be visible in the distribution types list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. Select the appropriate option.

The procedure 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.

Distributed Analysis Configuration - Options Tab

On the **Analysis Configuration** window, the **Options** tab allows you to specify options for the current analysis configuration.

> **Note:**
>
> Different design types have different options.

These options settings will be in effect only when all the following are true:

- A design is being solved whose design type matches the analysis configuration design type.
- The analysis configuration is the active configuration for its design type.
- You have not specified corresponding `batchoptions` on the command line.

Relation to Batchoptions

Command line `batchoptions` can be used to override the options specified by the active configuration.

Analysis configuration option settings can be overwritten by specifying the option name and value inside a `-batchoption` string.

To add batchoptions from the UI, click **HFSS 3D Layout** > **Submit Job**. Under **Analysis Options**, click **Add**. This opens the **Add Batchoption** window.

See: Running Electronics Desktop from the Command Line and Batchoptions Command Line Examples.

Adding Configurations or Accepting Edits

Click **OK** to accept the changes and close the **Analysis Configuration** window. Only machines marked **Enabled** appear in the machine list.

Regardless of the machine(s) on which the analysis is actually run, the number of processors and RAM Limit (%) settings, and the default process priority settings are now read from the machine from which you launch the analysis. See: Setting HPC and Analysis Options.

Selecting Optimal Configurations for Distributed Analysis

This section provides some basic guidelines for configuring a distributed analysis In this section, assume that a fixed number of machines and cores are available, and that the goal is to use them as efficiently as possible.

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 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.

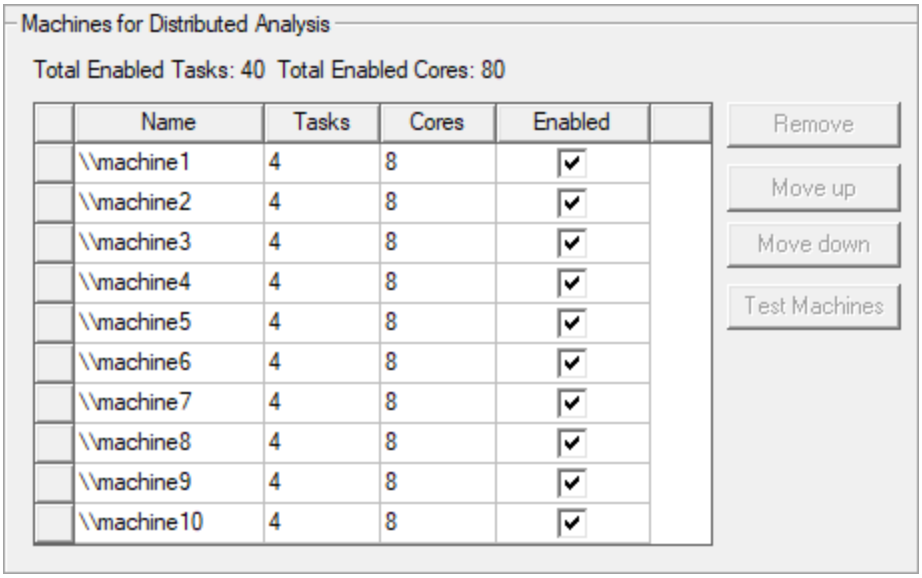
> **Note:**
>
> This topic assumes that there is enough hard drive space and memory for DSO simulations. For multiple DSO simulations on a single machine, the total memory needed is the sum of the memory used by each simulation. For example, consider a discrete frequency sweep in which each frequency 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.

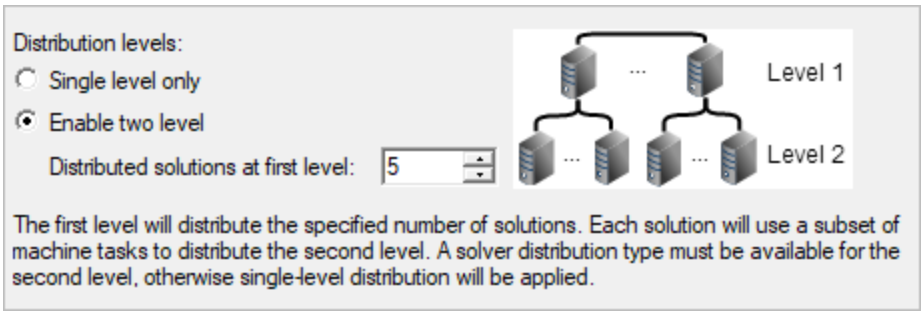
Two-Level Distribution Guidelines

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 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.

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 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`

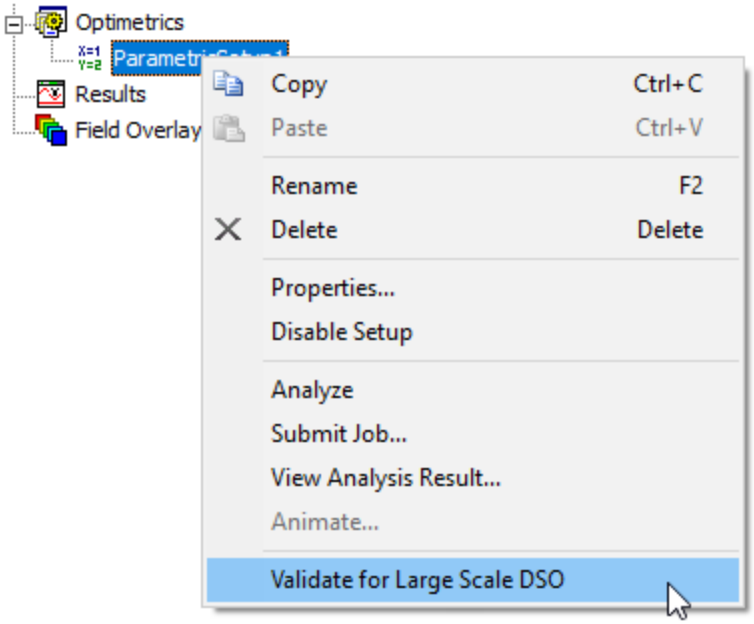
Job Management Interface for Large Scale DSO

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.

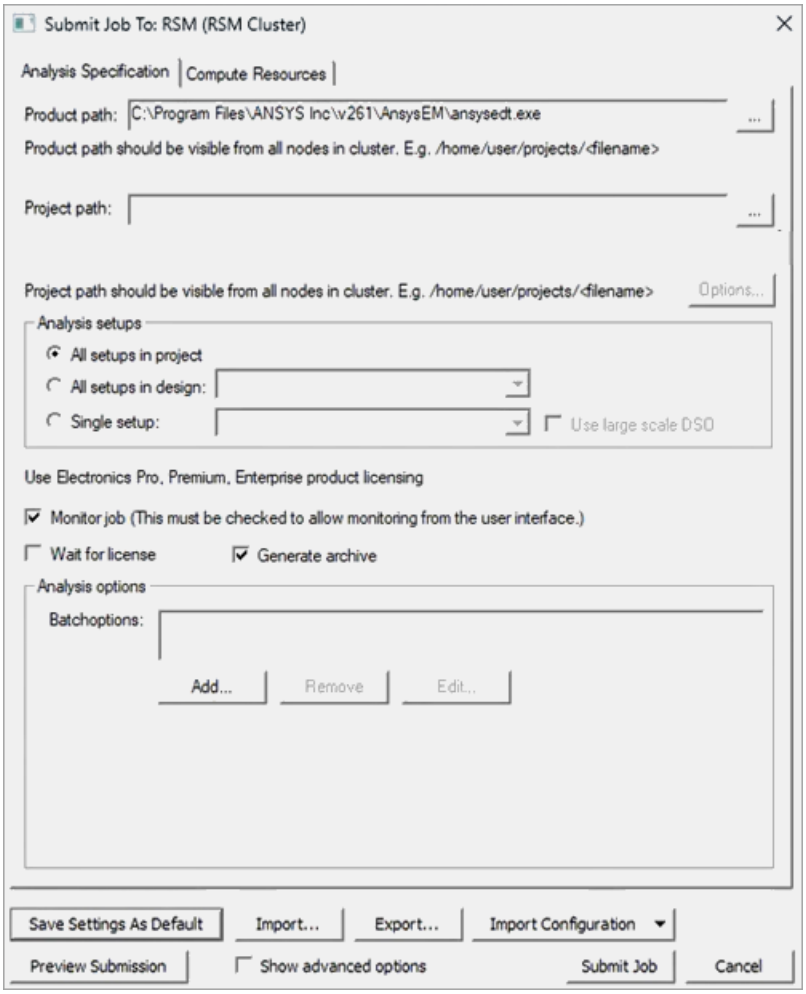
   In the RSM environment, you must specify a machine list (See: HPC and Analysis Options).

   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 **HFSS 3D Layout** > **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.

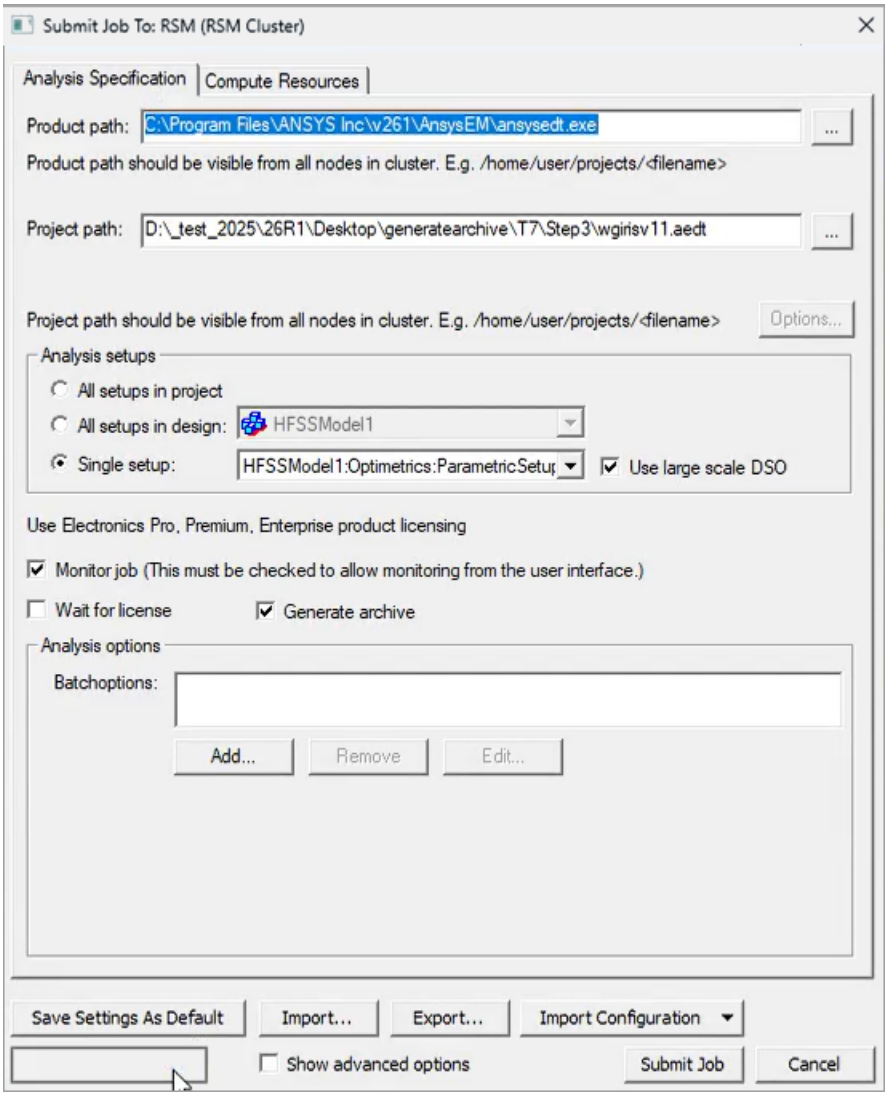
      For Large Scale DSO, select a **Single setup** from the drop-down menu and ensure the **Use large scale DSO** check box is selected.

    - 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**.

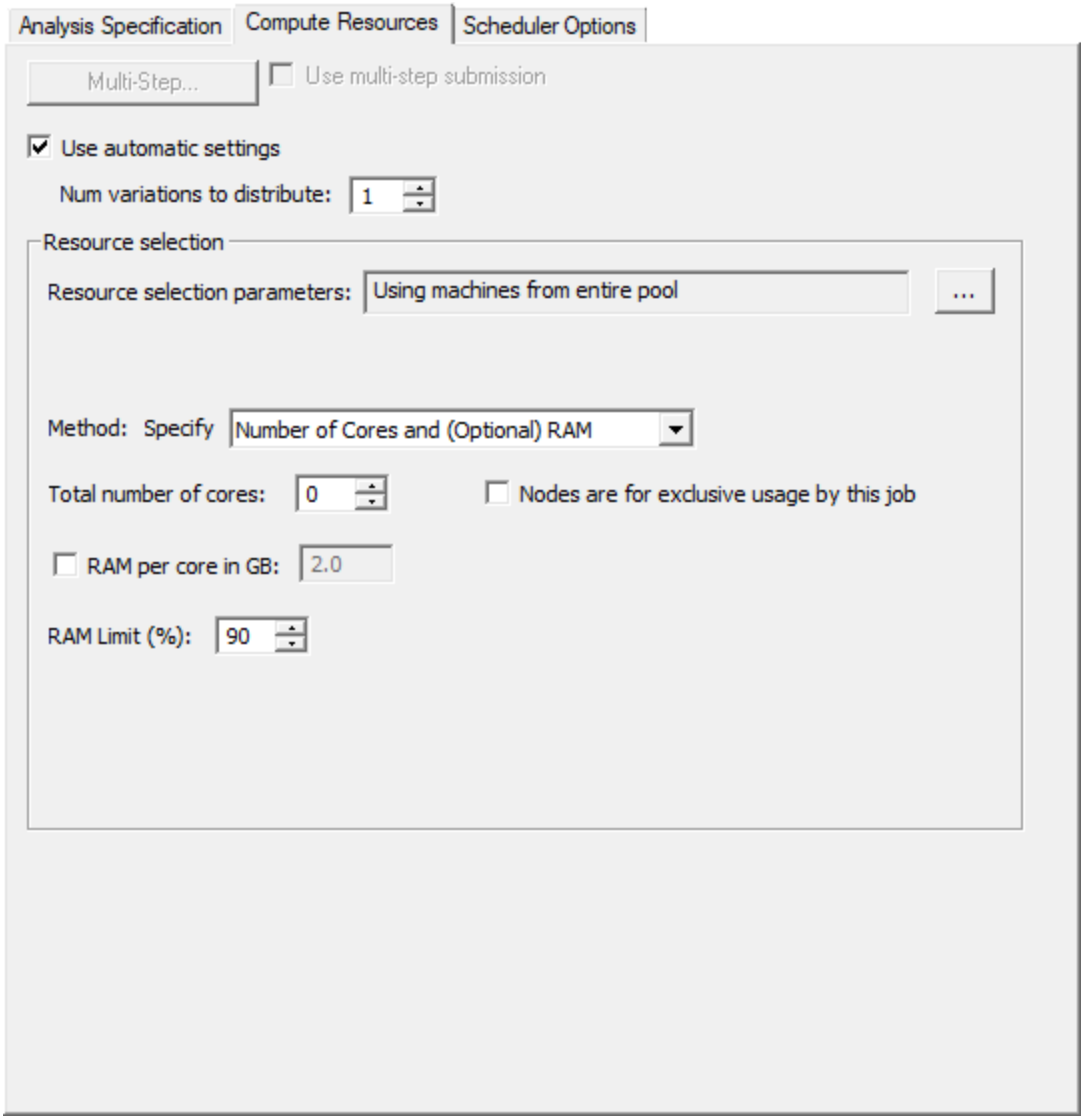
An example configuration is shown below:



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**.

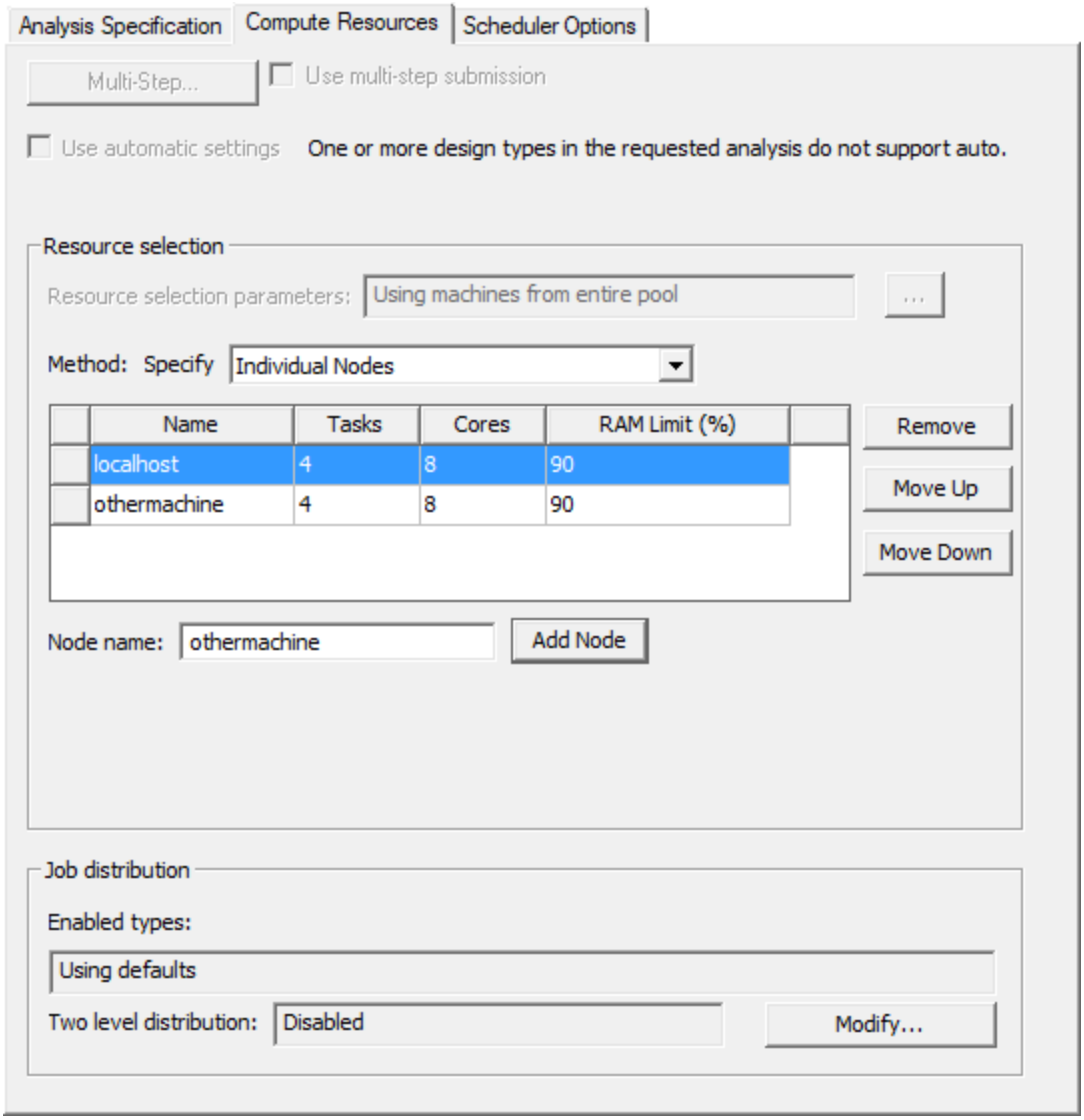
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



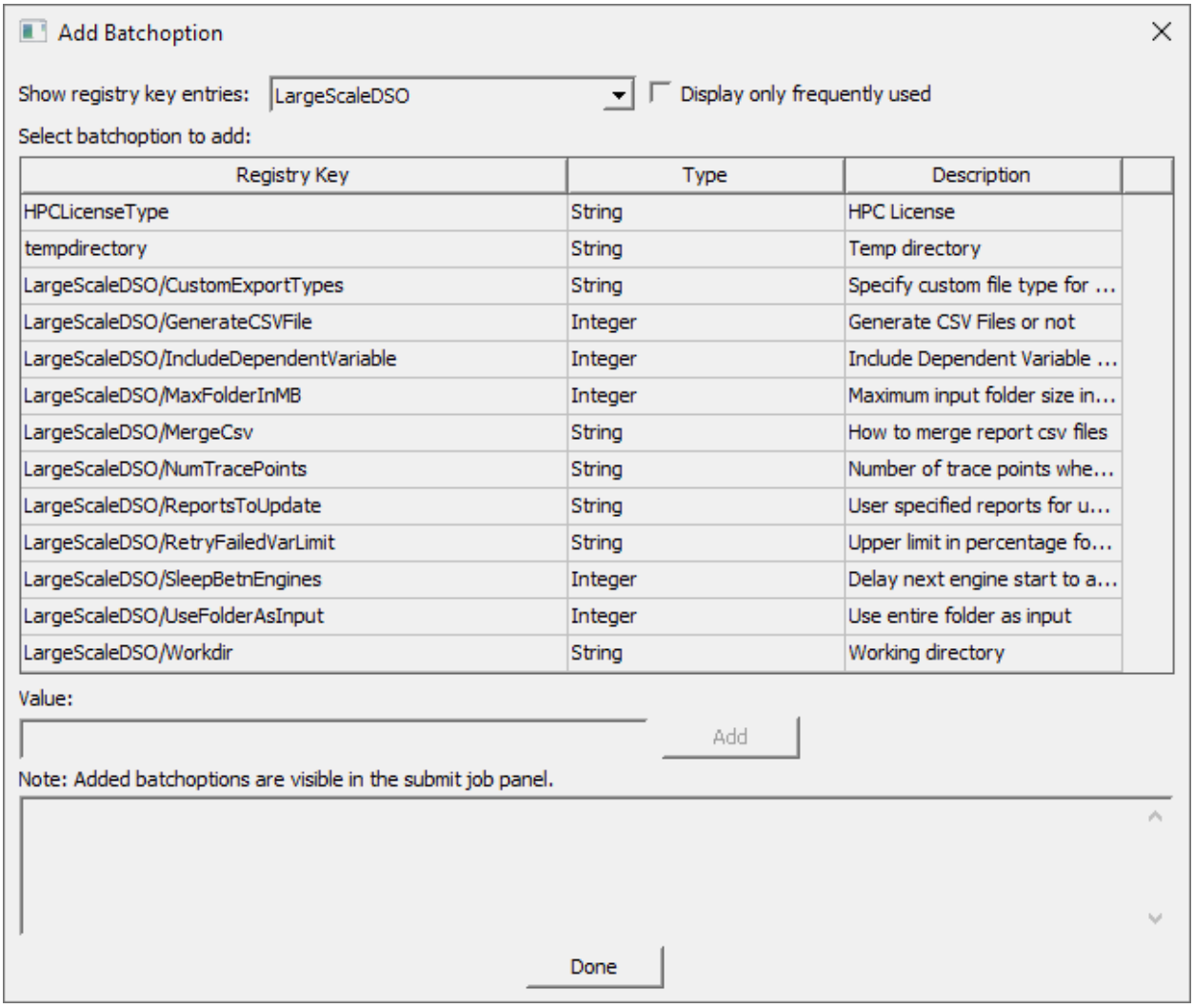
For RSM Large Scale DSO jobs submitted from the Job Submission panel, localhost must be the first node in the resource selection panel. Otherwise, the job will fail.

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.

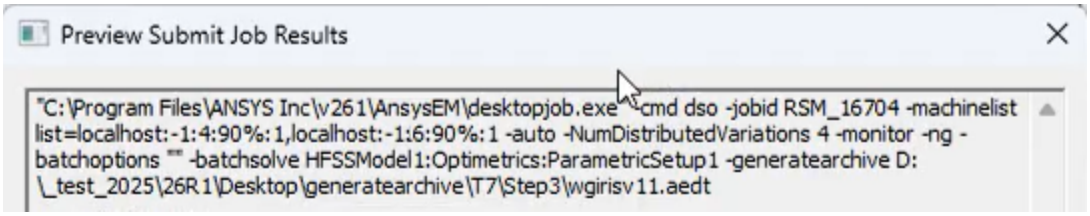
- **-batchsolve**

  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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

containing values for second value of theta. The magrE output columns are titled as 'magrE_crv1' and 'magrE_crv2' respectively.

- **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.

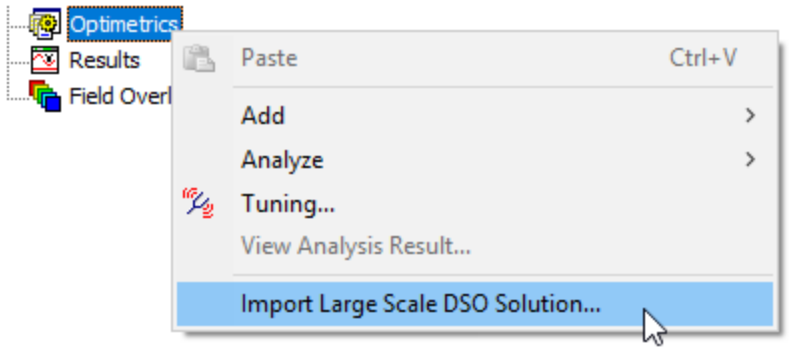
Postprocessing Large Scale DSO Dataset Solutions

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:

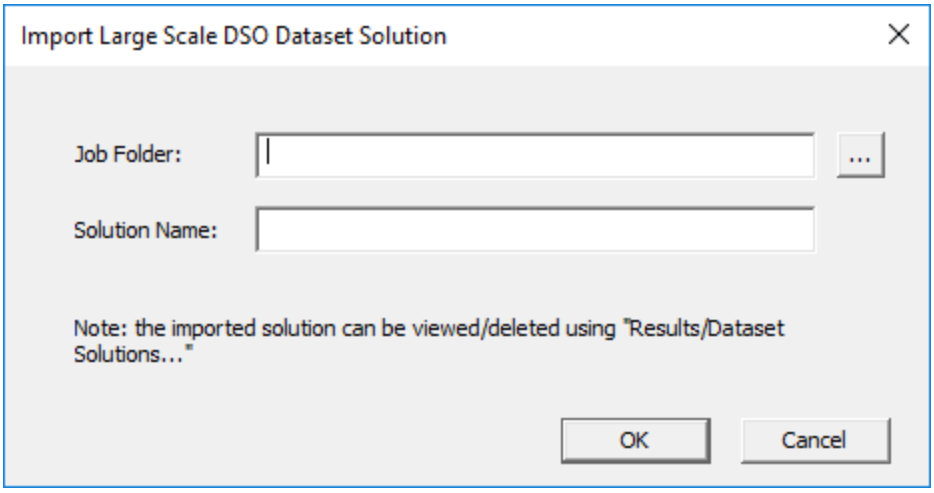
1. In the Project Manager, right-click **Optimetrics** and select **Import Large Scale DSO Solution**.



> **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.

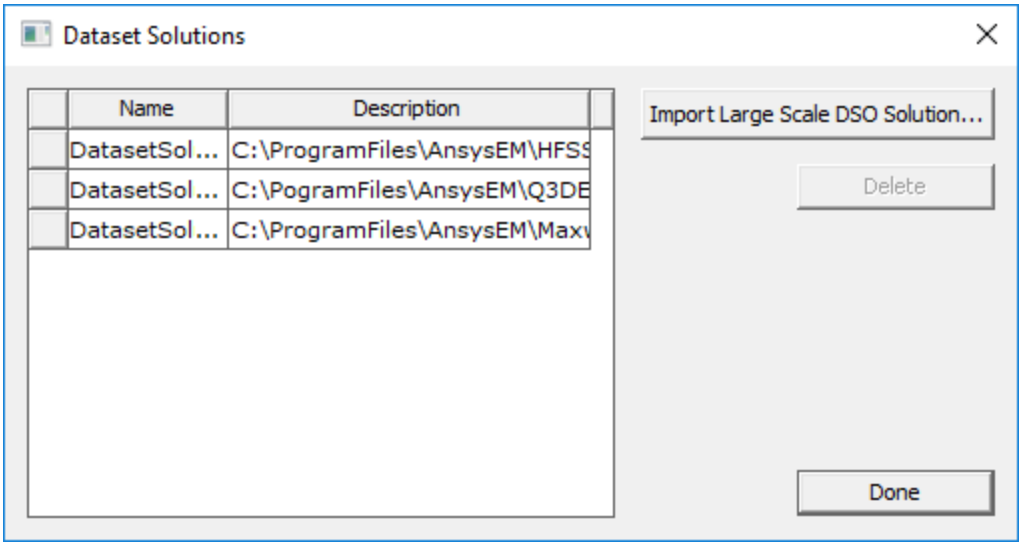
Viewing Imported Solutions

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.

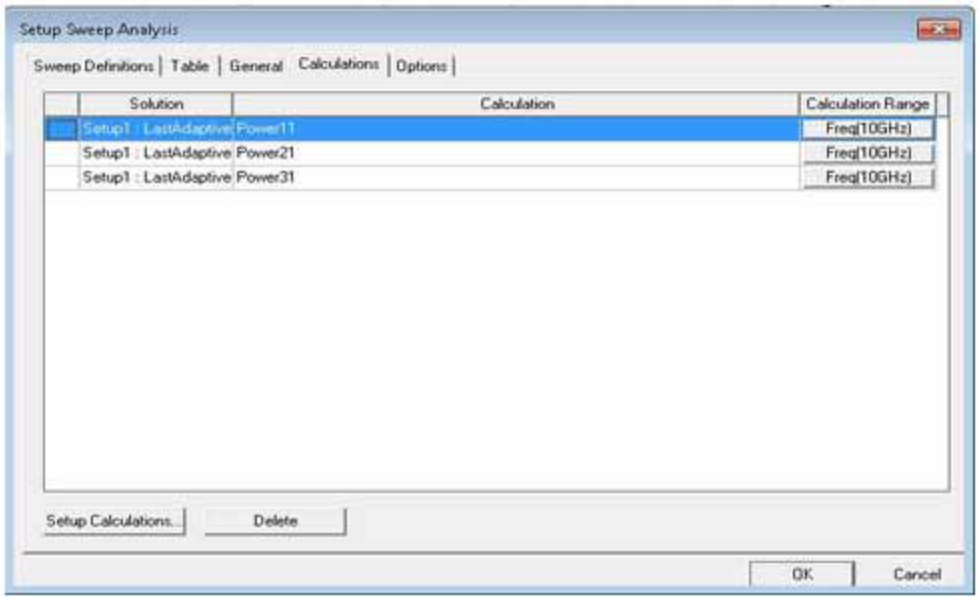
   For this example, use the existing 'ParametricSetup1' as the parametric setup to solve.

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.

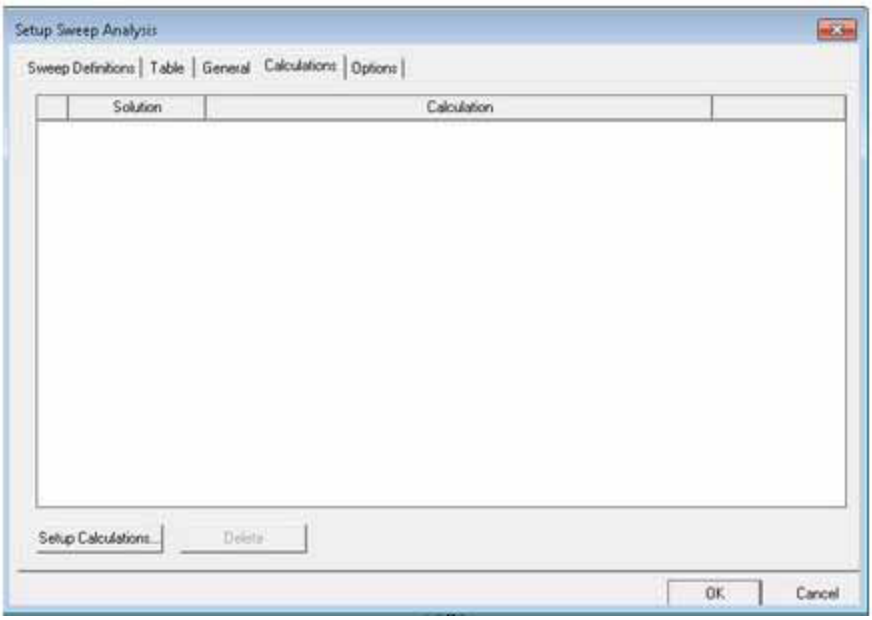
   a. Because these DSO outputs come solely from Rectangular Plots, delete all other postprocessing setups, and then turn off Save Fields And Mesh as shown below.

      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.



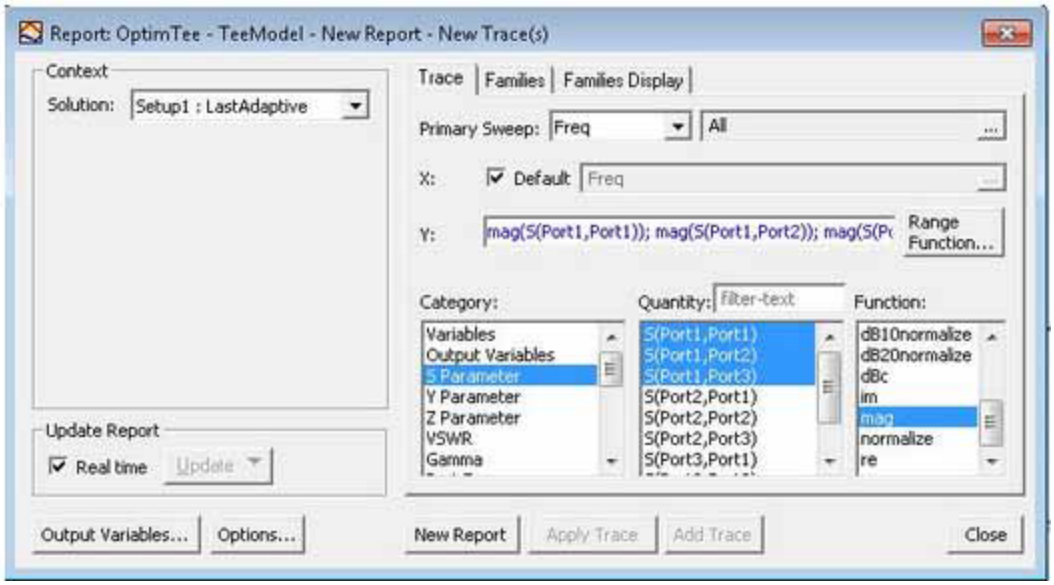
c. Click on the **Options** tab and uncheck Save Fields And Mesh, as shown below:

5. Use the Reporter to define outputs.

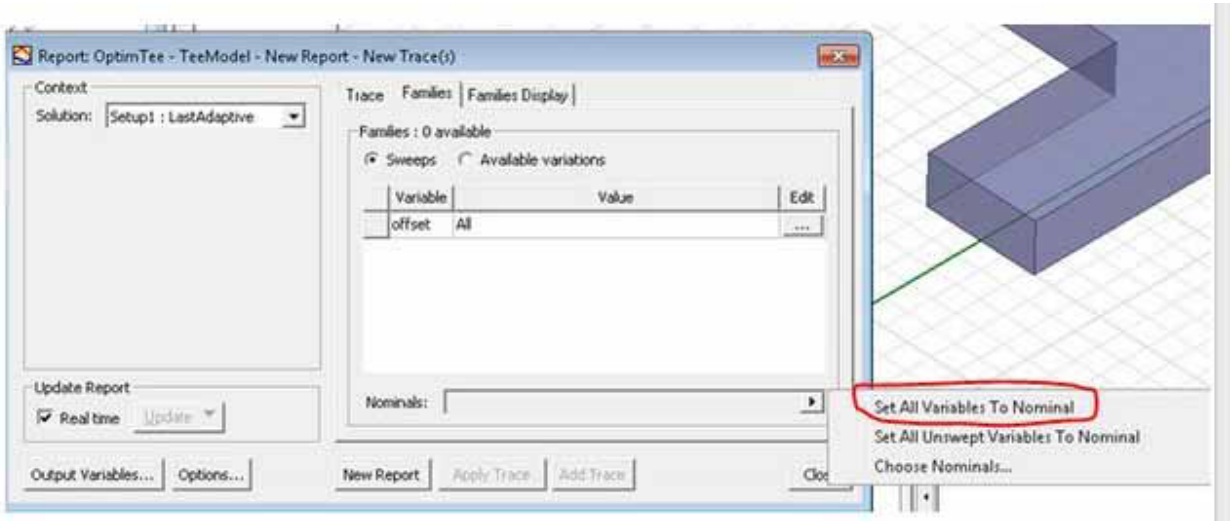
   For this example, you add six traces that correspond to six csv outputs of the Large Scale DSO job: mag(S11), mag(S12), mag(S13), Power11, Power12, and Power13.

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.

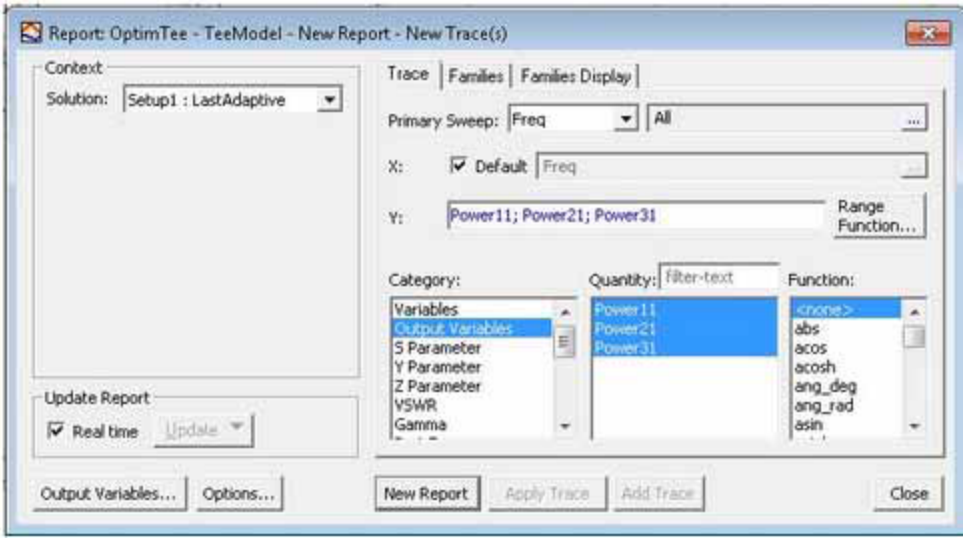
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

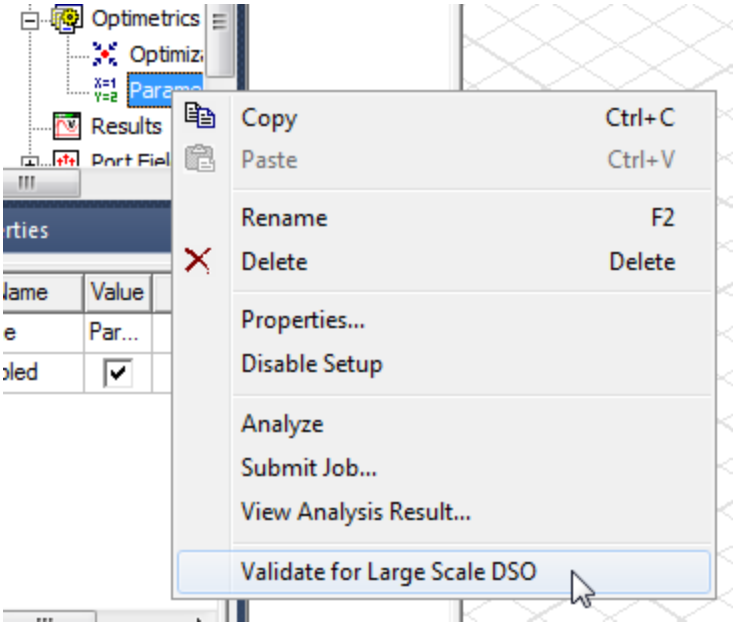


f. Click on the **Families** tab and ensure that all variables are set to **Nominal**.

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 " 'HFSS 3D Layout Design/NumCoresPerDistributedTask'=11 'HFSS 3D Layout`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Design/RAMLimitPercent'=90 '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 " 'HFSS 3D Layout Design/RAMLimitPercent'=90 '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 " 'HFSS 3D Layout Design/RAMLimitPercent'=90 'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

Next

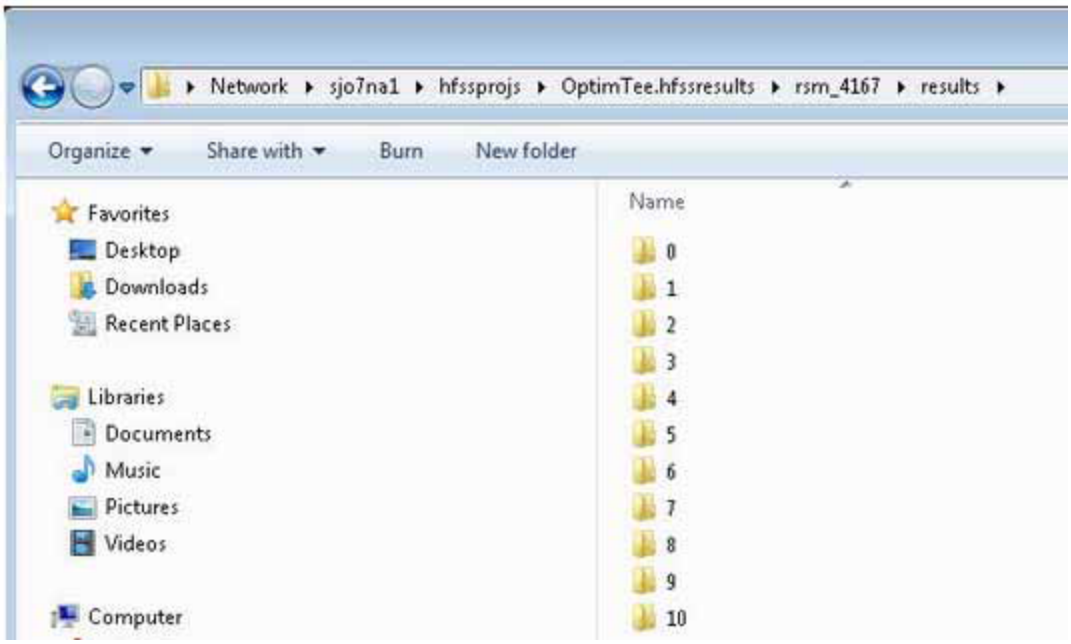
Large Scale DSO Example: Post Process the Results

Large Scale DSO Example: Post Process the Results

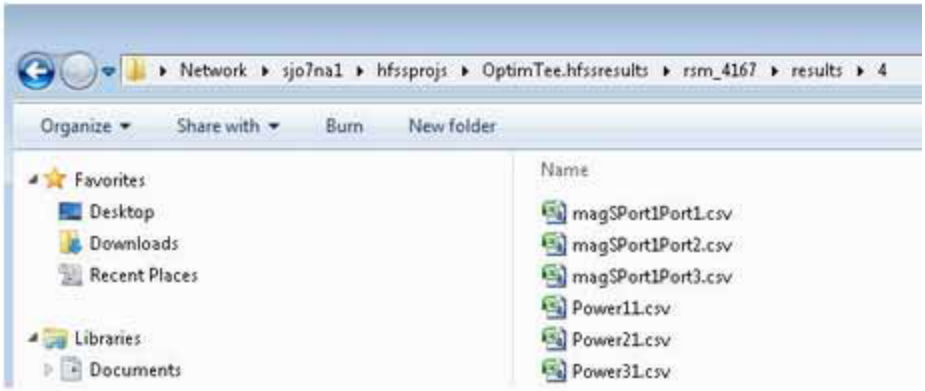
Once the job is done, output is available in the ~\OptimTee.aedt\jobid\results folder. Each variation creates a subfolder, which in turn has one csv file per trace of each report. See the detailed information regarding job monitoring and the location of the analysis logs.

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.

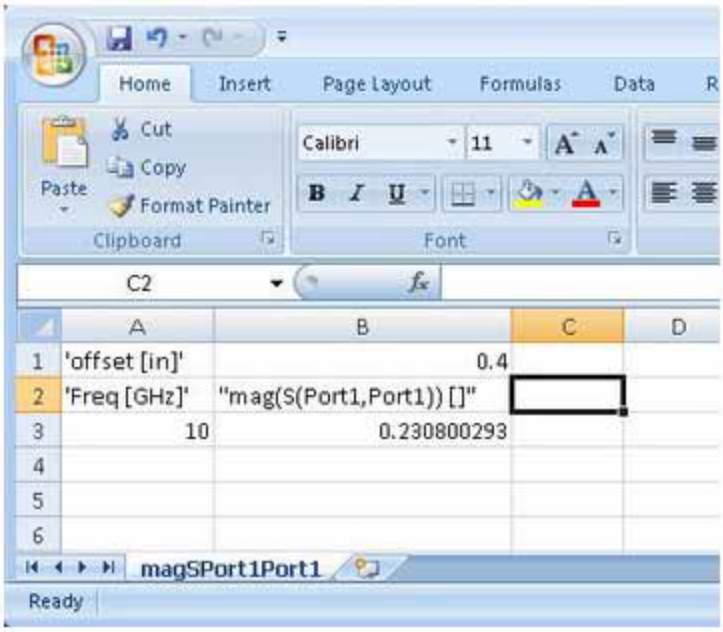


You have three options for postprocessing csv files.

- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

allocated to the job by the scheduler will be used for the job. In general, the job distribution 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.
- **RAMLimitPercent:** Specifies the percentage of system RAM that the analysis is expected to use on each host. Running out of memory can cause processes to be killed by the Operating System or other failures. The analysis will be gracefully terminated if it requires more than this limit. Typical limits are about 90%, to allow some RAM for the Operating System and other services. If there are multiple jobs running on a single host, then the RAM limit should be reduced accordingly. For example, if there are two jobs running on a host, then the limit could be 45%, which would allow 10% of the RAM for the Operating System and other services.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **RAMLimitPerCoreInGB:** This is an alternative way to specify the RAM limit, instead of using the RAMLimitPercent. In this case, the RAM limit is the limit per core multiplied by the number of cores allocated to the job on each host. Typically used for jobs running under a scheduler to ensure that the scheduler memory limit and the ansysedt memory limit are consistent.
- **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.

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 "
```

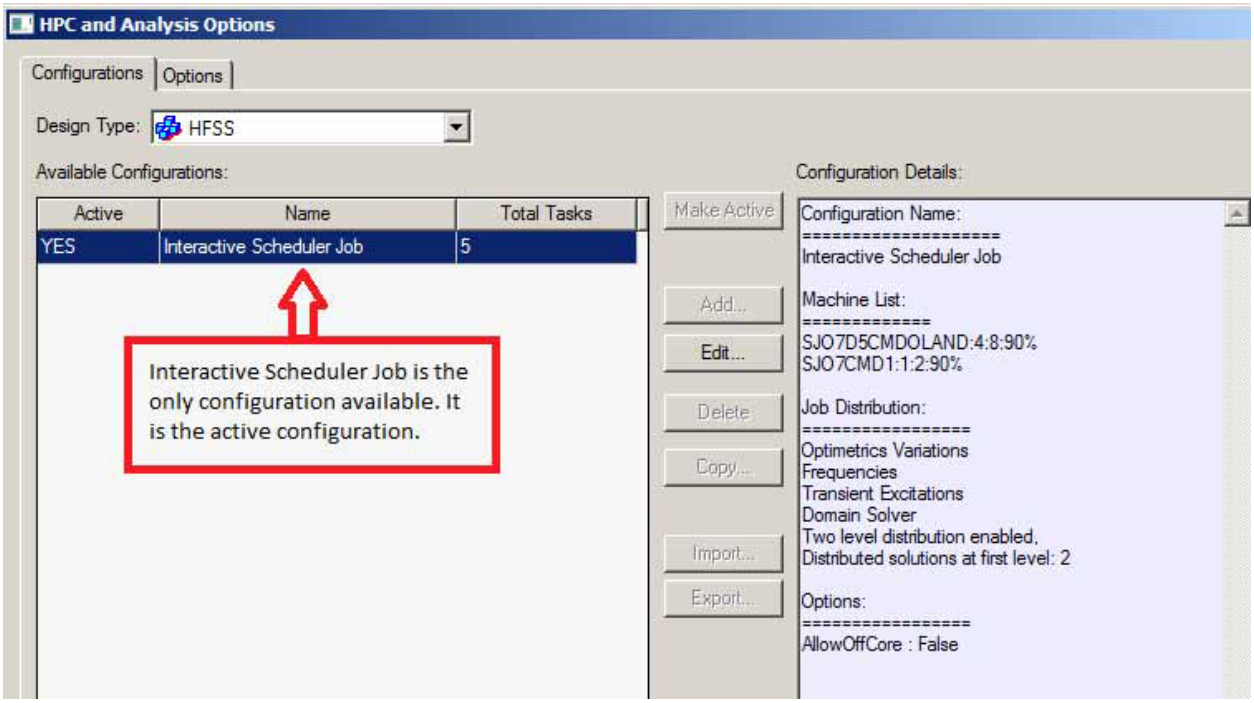
DSO Configuration for Interactive Scheduler Jobs

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



One major difference between interactive scheduler mode and batch mode or normal interactive mode is:

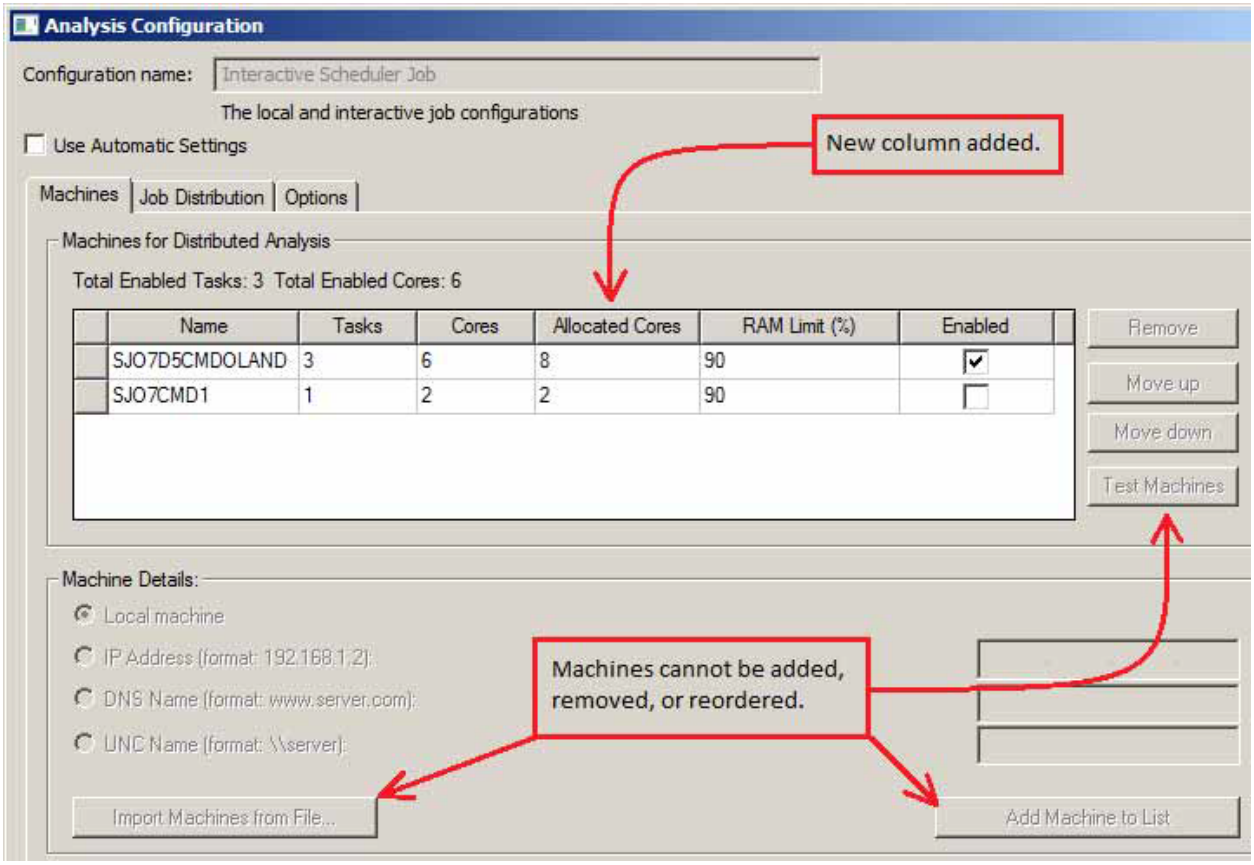
- In interactive scheduler mode, DSO configuration settings and design type options in the UI override command line options and batchoptions.
- When not in interactive scheduler mode, DSO configuration settings and design type options specified on the command line (including batchoptions) override UI 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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, cores, and RAM limit for any machine, or specify that a machine is enabled or disabled.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

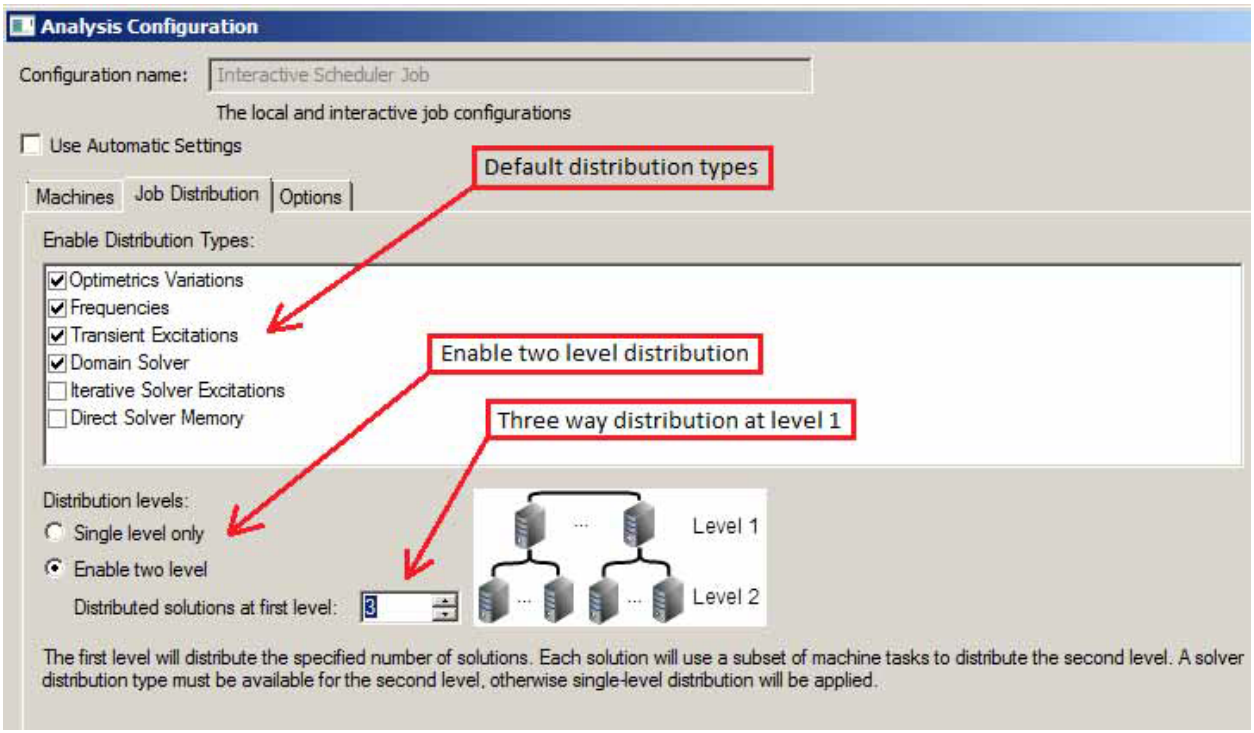
The registry contains the last value of these settings for an interactive scheduler job for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. 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 -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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

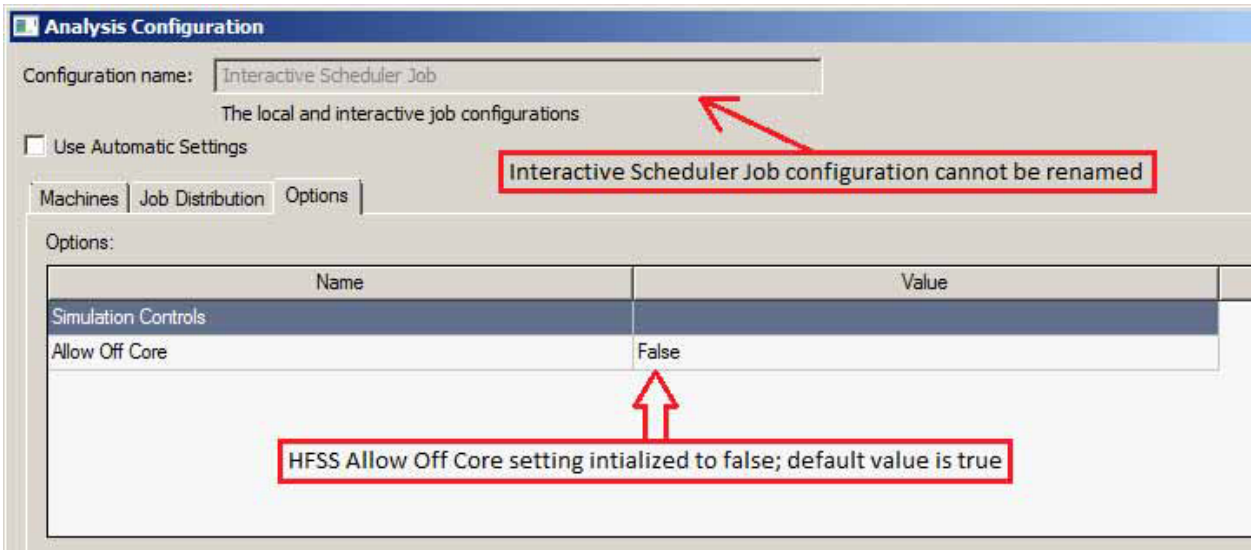
override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the "Interactive Scheduler Job" configuration for the current design type.

Example

Command line:

```
ansysedt -batchoptions 'HFSS/AllowOffCore'=0
```

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

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 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'
```

Distribution Command Line Options

The user should include options specifying how the job should be distributed in the command line. These command line options will override options specified using the **HPC and Analysis Options** dialog box or obtained from the registry. See Running Ansys Electronics Desktop from a command line for more information on command line options.

For both scheduler batch jobs and interactive scheduler jobs, the -MachineList num=<num distributed tasks> format is the most common way to specify the number of tasks for the job. The other formats (-MachineList list=... or -MachineList file=...) allow the user to specify the number of tasks and cores to use on each host. These formats may be useful with clusters of heterogeneous machines, by allowing the user to specify different numbers of tasks or cores for different hosts. If either of the latter two formats is used, the user must ensure that the hosts and cores specified on the product command line are compatible with the hosts and cores allocated to the job.

Batchoptions for Interactive Scheduler Jobs

For interactive scheduler jobs, only a limited set of batchoptions are supported. These batchoptions include the DSO configuration options, the design-type-specific options, and the following additional design-type-specific options that are not currently listed in the user interface or command line help windows:

- **CreateStartingMesh:** Create the starting mesh only. Do not solve any other steps. Often used for multi-step jobs where the mesh creation is solved in a separate step using fewer resources than the rest of the analysis.
- **NumCoresPerDistributedTask:** Specifies the number of cores that are allocated to the job to use for each distributed task, used when running jobs under a scheduler. The scheduler communicates the number of cores allocated to the job on each host. This setting is used to determine how to allocate the cores to each task.
- **RAMLimitPercent:** Specifies the percentage of system RAM that the analysis is expected to use on each host. Running out of memory can cause processes to be killed by the Operating System or other failures. The analysis will be gracefully terminated if it requires more than this limit. Typical limits are about 90%, to allow some RAM for the Operating System and other services. If there are multiple jobs running on a single host, then the RAM limit should be reduced accordingly. For example, if there are two jobs running on a host, then the limit could be 45%, which would allow 10% of the RAM for the Operating System and other services.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **RAMLimitPerCoreInGB:** This is an alternative way to specify the RAM limit, instead of using the RAMLimitPercent. In this case, the RAM limit is the limit per core multiplied by the number of cores allocated to the job on each host. Typically used for jobs running under a scheduler to ensure that the scheduler memory limit and the ansysedt memory limit are consistent.
- **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 the Ram Limit Per Core in GB

When cores and RAM per core are requested, the cores could be allocated in an arbitrary/non-uniform way across nodes that themselves could be non-uniform/heterogeneous. For example, nodes could range from 4 cores to 20 cores and from 64 GB to 384 GB. In such environments, the RAM percentage set in the HPC configuration, Job Submission Compute Resources, or batch option may not be appropriate. In such cases, you can use the batchoption for RAMLimitPerCoreInGB. This setting specifies the Maximum amount of RAM used for each core allocated by the scheduler in GB. This batchoption cannot b e combined with RAM limit percent and is only valid when solving in a Linux scheduler environment.

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".

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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
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.
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.

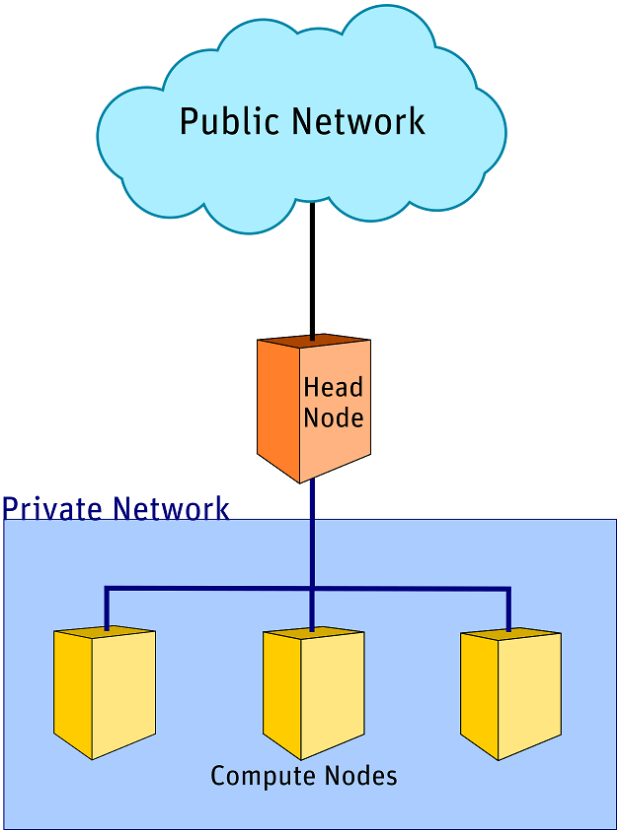
What a Scheduler Does

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.

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.

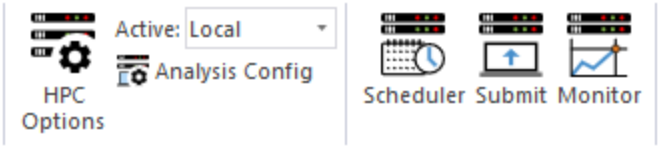
Job Submission User Profile on Cluster Compute Nodes

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

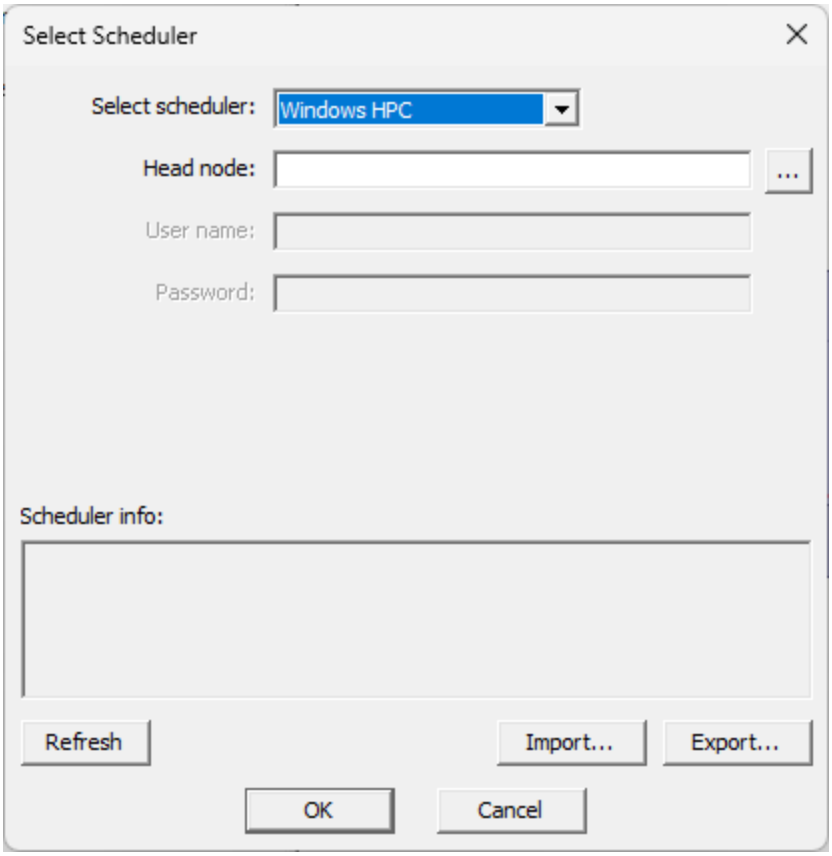
In order to submit jobs using **Windows HPC**:

- 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 **HFSS 3D Layout** > **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.

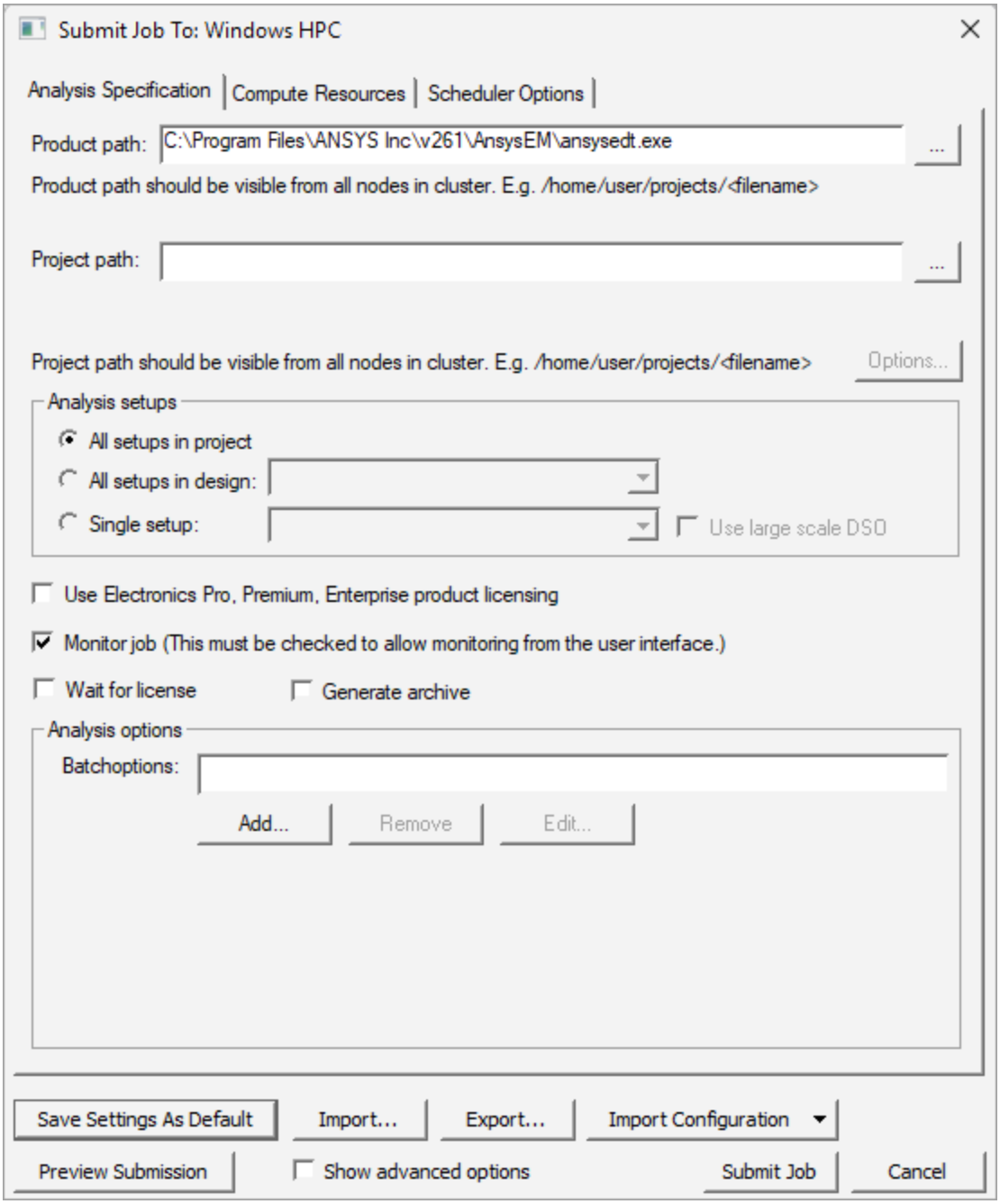
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

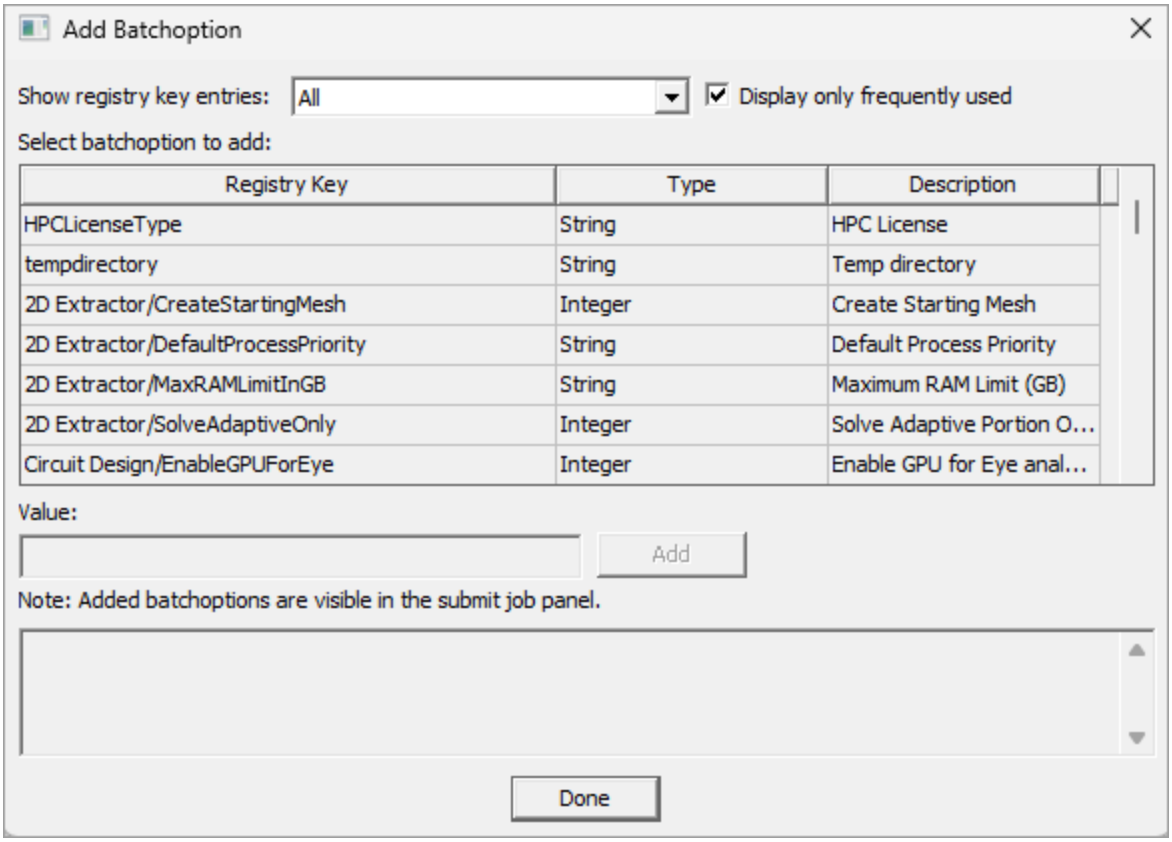
- **Monitor job** – you must enable this option to monitor the job from the user interface. You can then monitor this job through the **Tools** > **Job Management** > **Monitor Jobs...** function or by checking the dialog box that opens when you submit the job.
- **Wait for license** – specify whether 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.



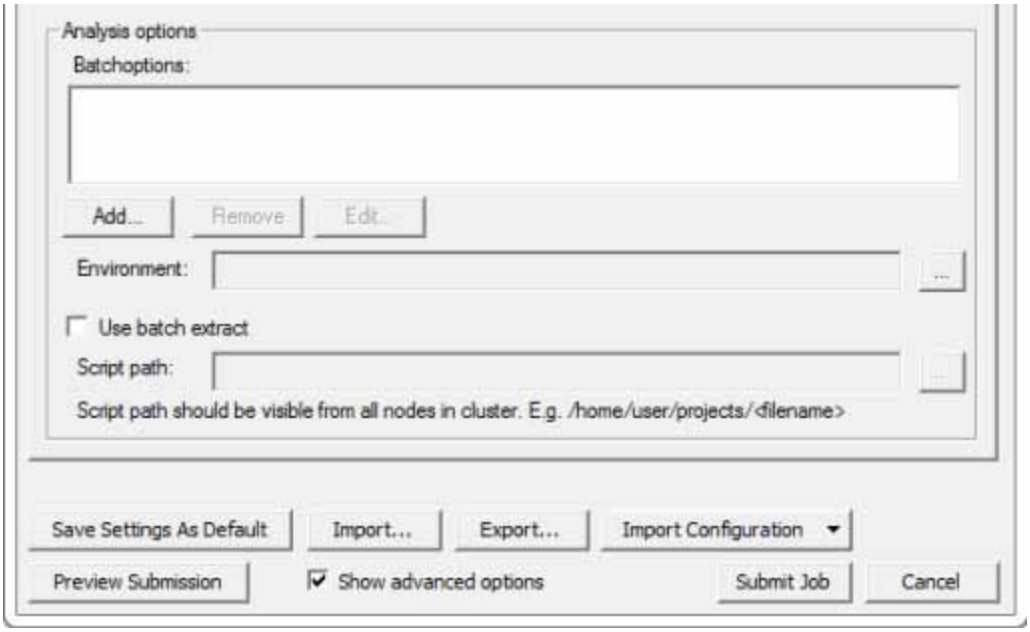
This dialog box provides access to all -batchoption commands. The drop-down menu lets you select specific categories, and you can choose to display only frequently used commands. You can edit and remove any batch options you specify.

Select a Registry Key in order to show the current Value for the type. The lower field explains the meaning of the Type Value.

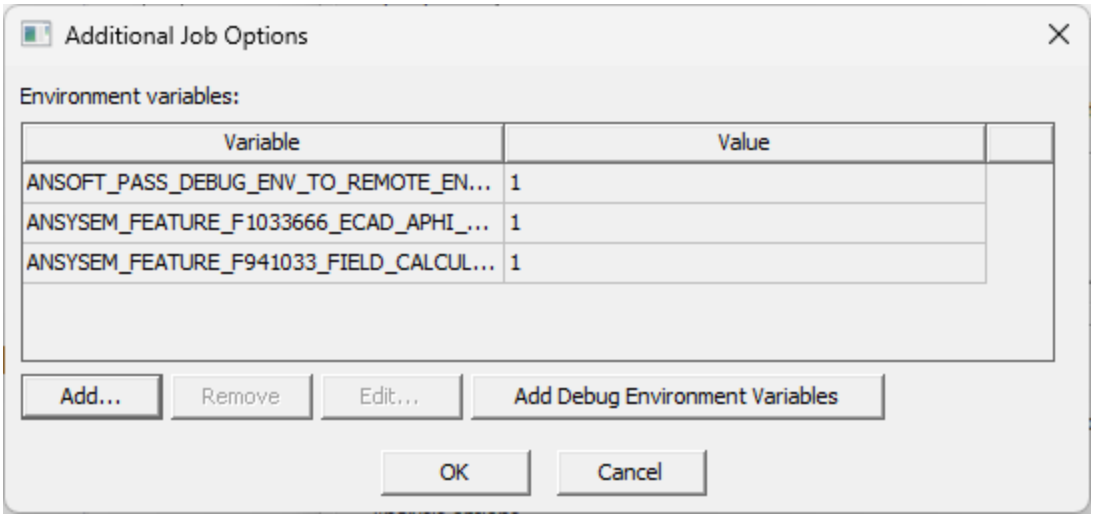
Any batchoptions for which you select **Add** will be visible in the **Submit Job** dialog box.

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.

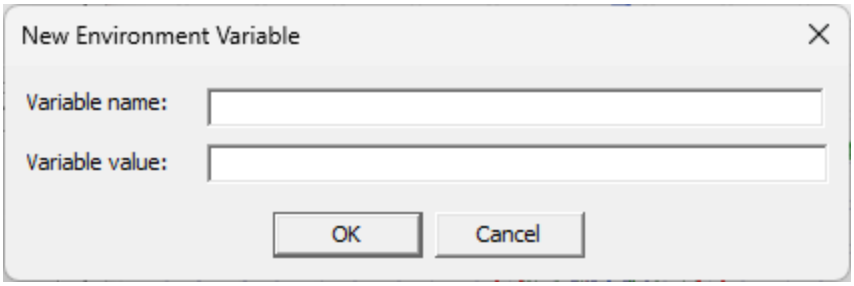


The **Environment** field lets you specify any environment variables. Click the ellipsis button [...] to display the **Additional Job Options** dialog box.



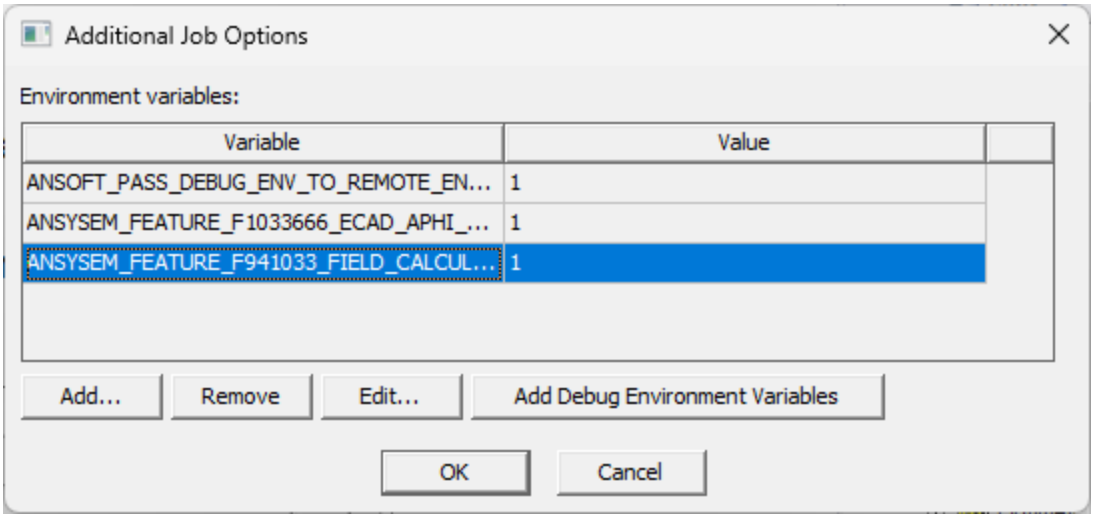
Click the **Add...** button to open the **New Environment Variable** dialog box.

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.

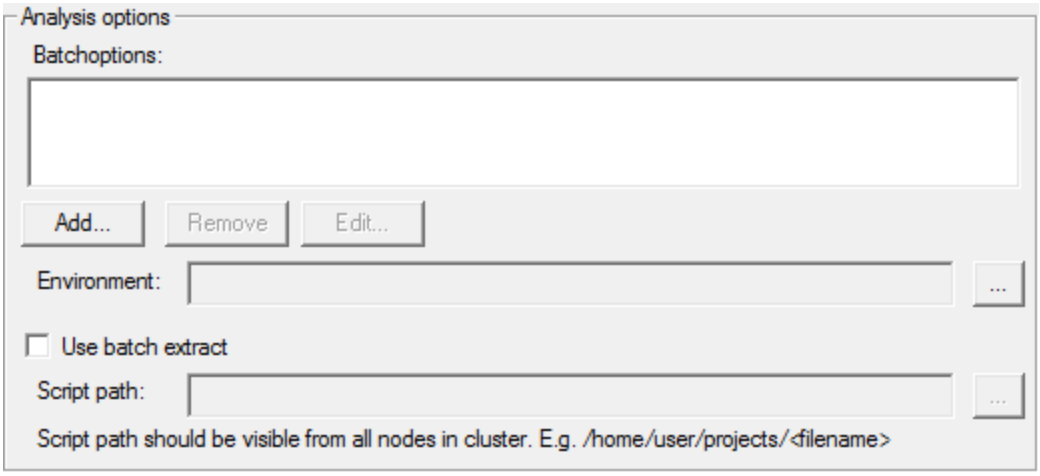
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

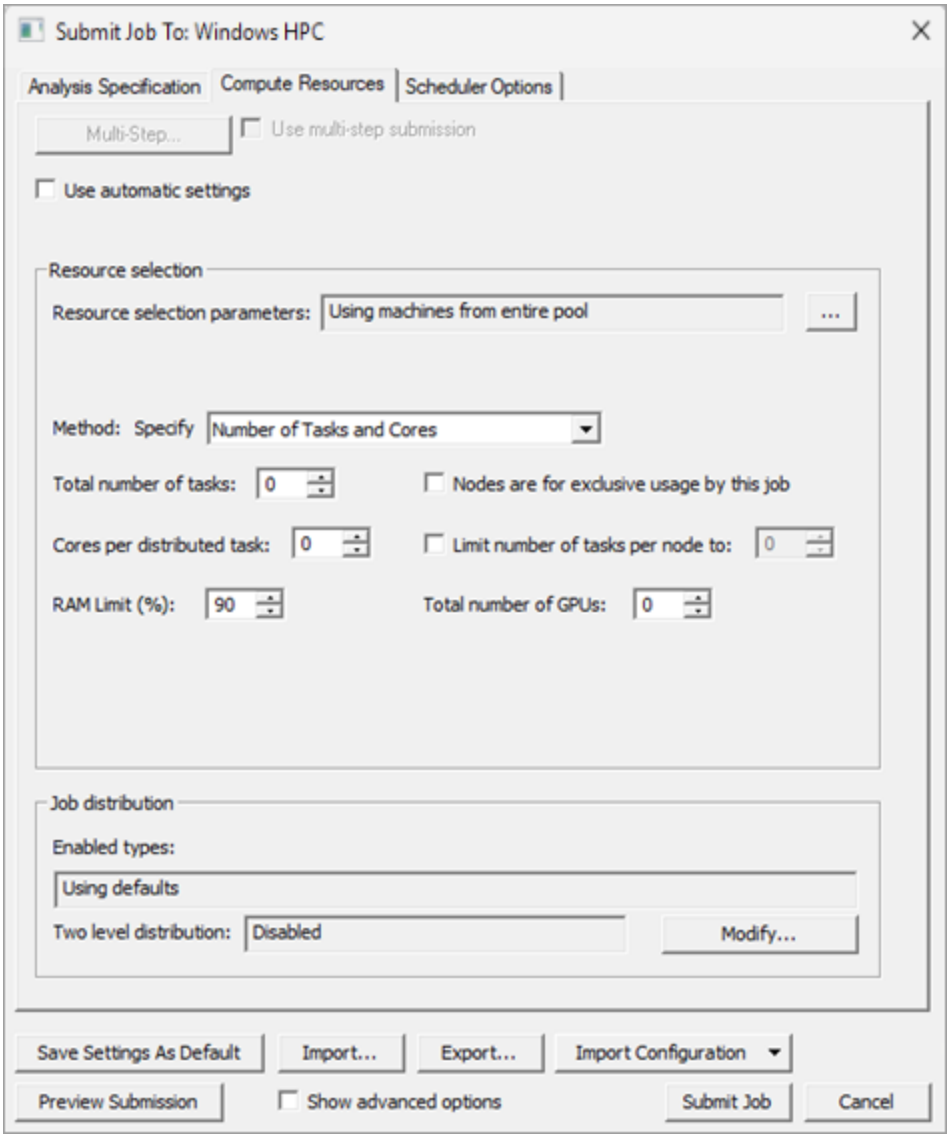


See Running Ansys Electronics Desktop from a Command line for an explanation of the solve information available through batch extract.

The **Preview Submission** button opens a window that shows the text commands that will be sent to the scheduler.

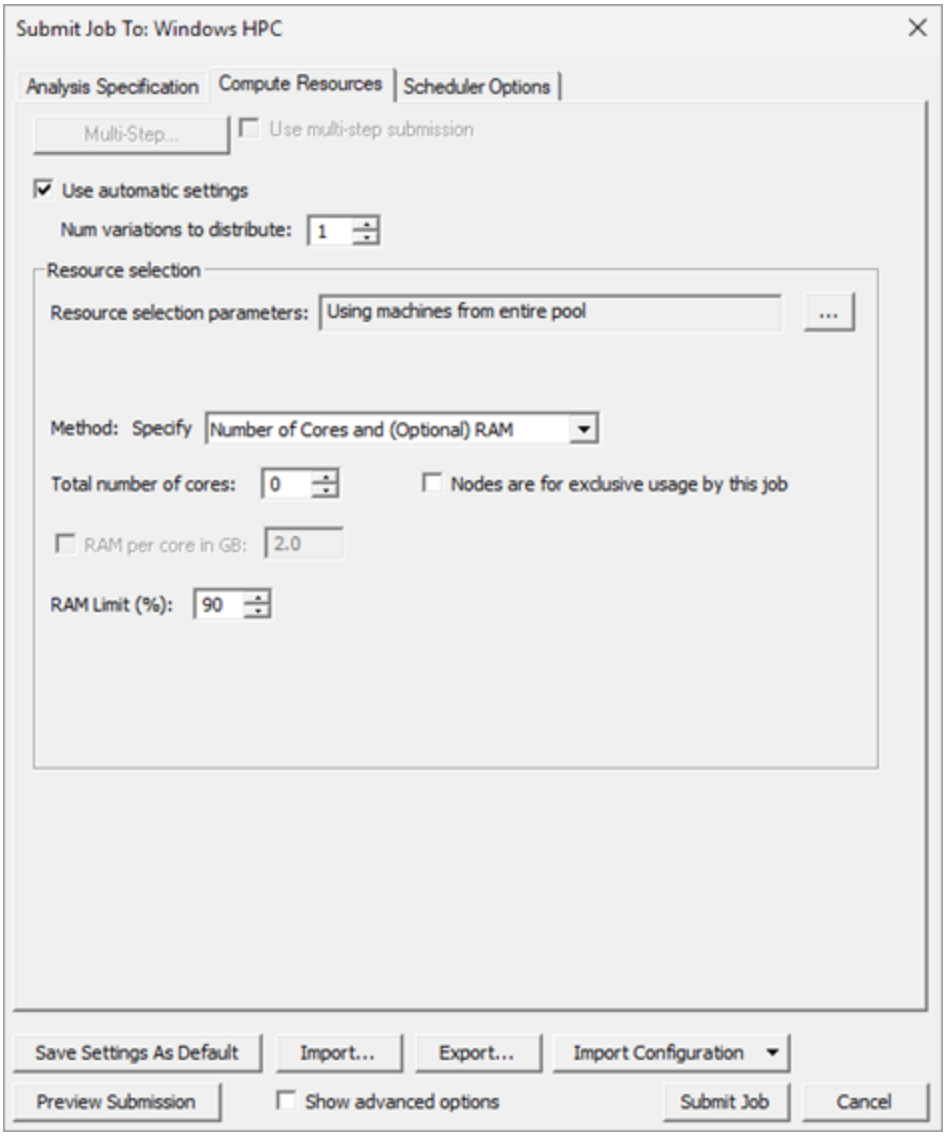
The following figure shows the **Compute Resources** tab of the **Submit Job To** dialog box.

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.

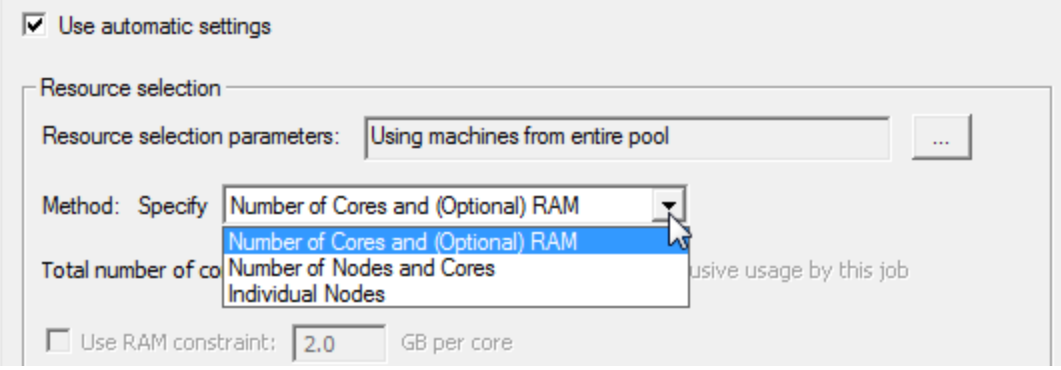
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



HFSS and HFSS 3D Layout have **Use automatic settings** selected by default. With **Use automatic settings** selected, the **Job distribution** field is removed. When using automatic settings, you can specify **Resource selection** parameters. The ellipsis button [...] opens the Compute Resource Selection Parameters dialog box. If you do not specify any parameters, the default is "Using machines from entire pool."

The Method field of the **Submit Job To** dialog box has a drop-down menu with two or three selections, depending on whether you select **Use automatic settings**.

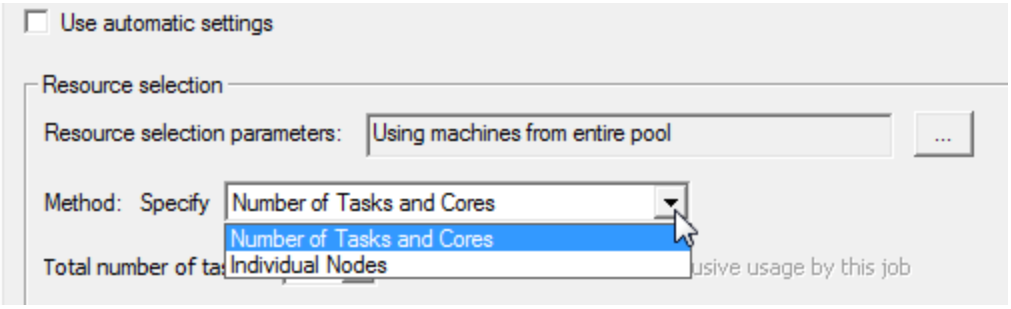
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

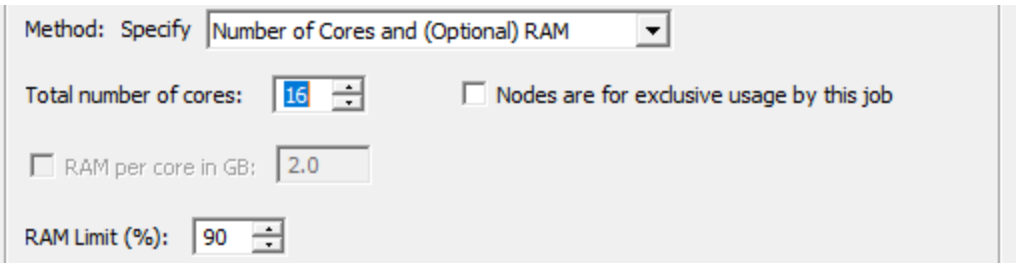
If you select **Use automatic settings**, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains.

If you uncheck or cannot access **Use automatic settings**, these two Methods are listed:



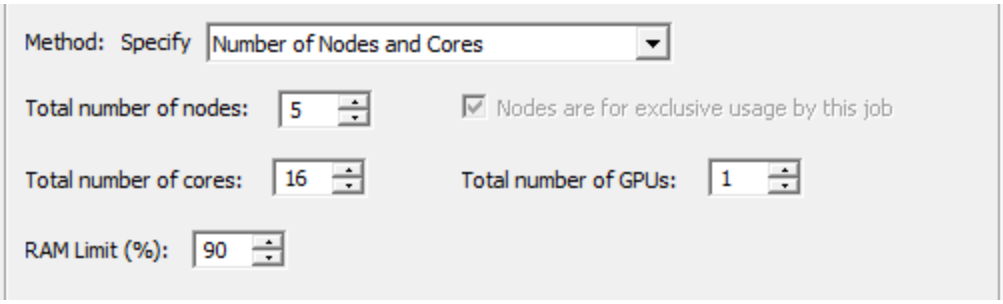
For Windows, there is a 64-core limit for a single machine. Each Method selection changes the available options listed:

- Specify Number of Cores and (Optional) RAM

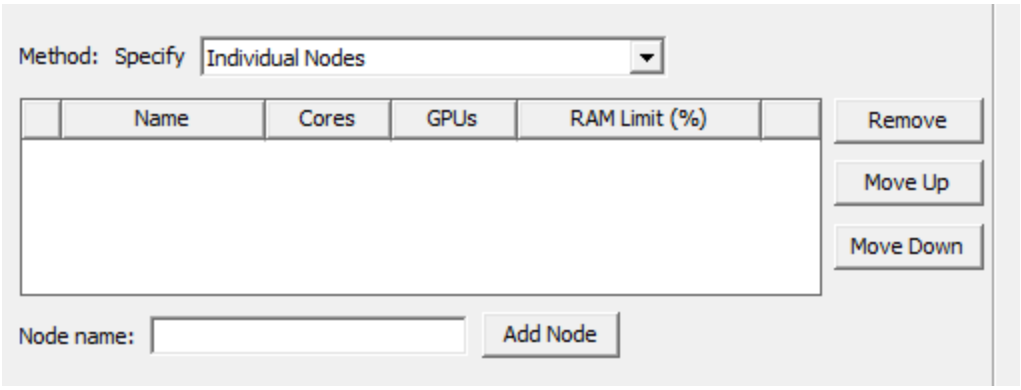


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Number of Nodes and Cores

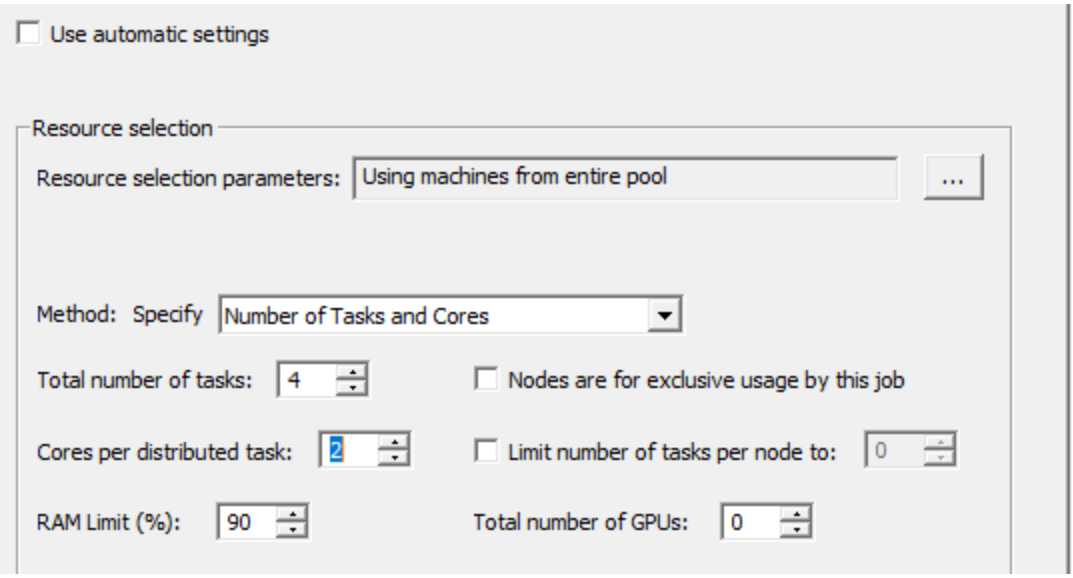


- Individual Nodes



- 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):

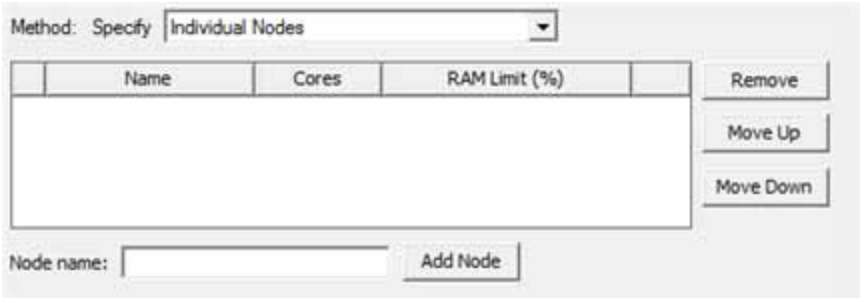
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Individual Node List

For Windows HPC jobs, you may either specify a node list, or specify job parallelization parameters, but not both.

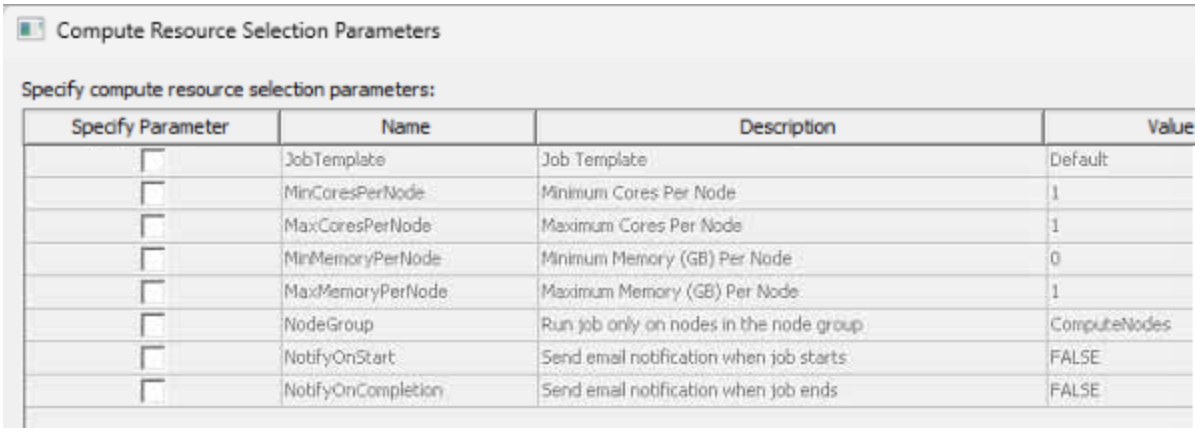
If you select the Individual Nodes Method, you may specify a node list, and the Job parallelization controls are disabled. In this case, the node list should only include cluster nodes that are valid for the job. For each node, you enter the node name and add the node. In the table, you can specify the 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

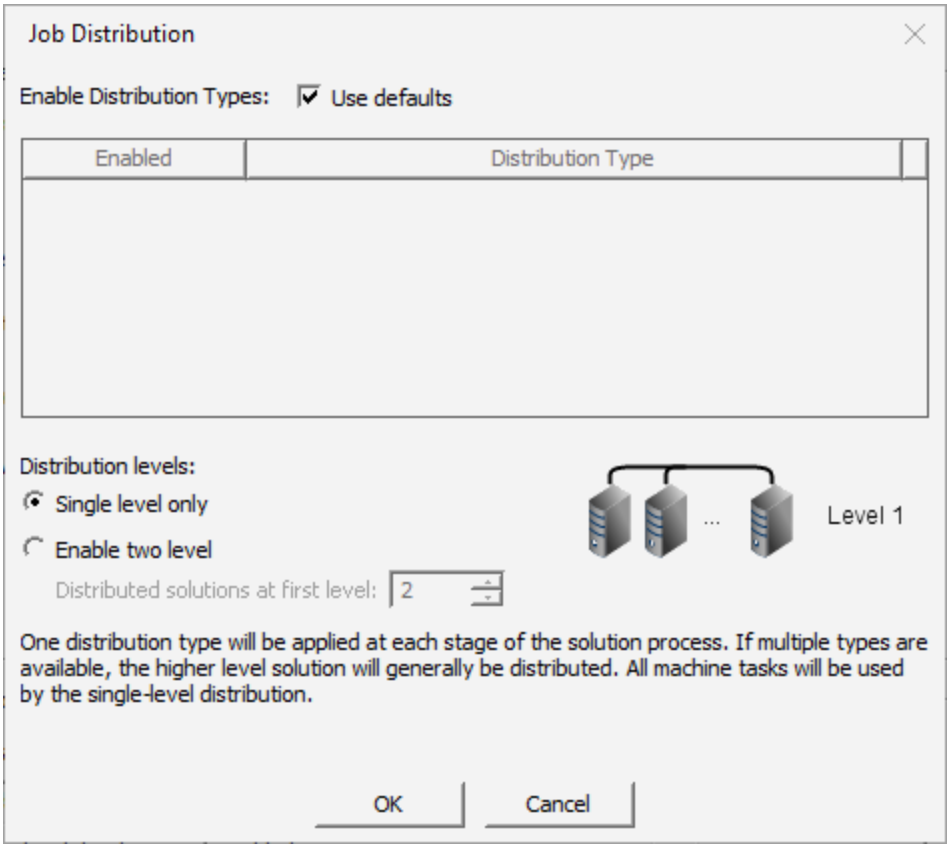
For Windows HPC jobs, you may either specify a node list, or specify the job parallelization parameters, but not both. The Job parallelization fields let you specify:

- 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".

Job Distribution

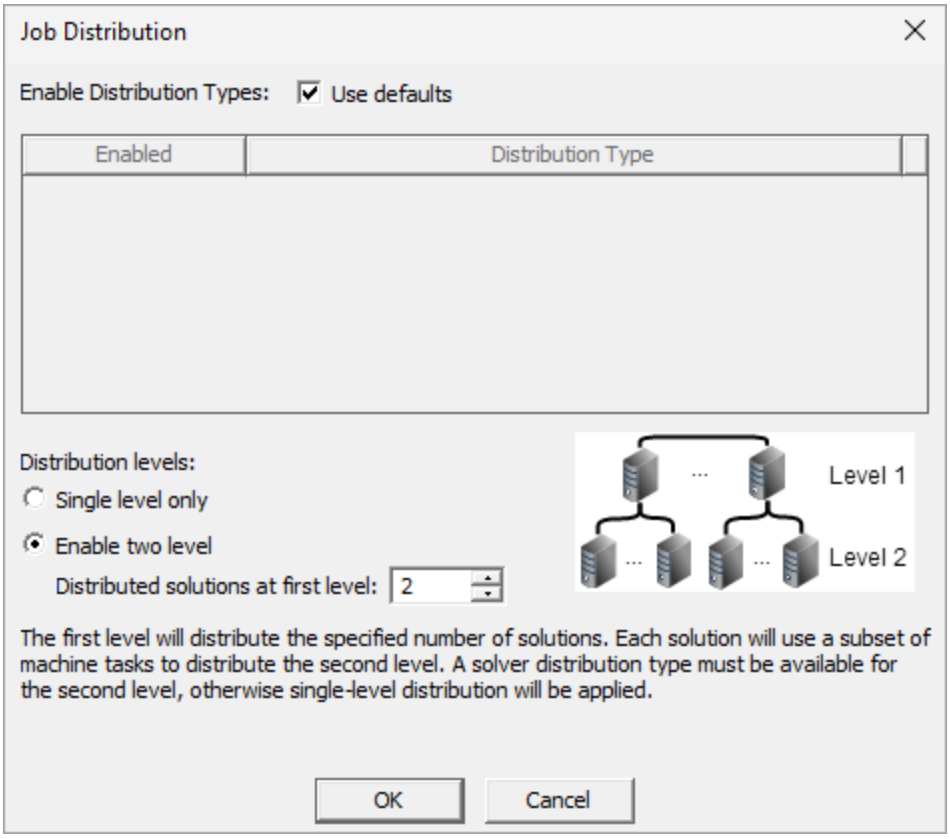
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 the number distributed solutions 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.

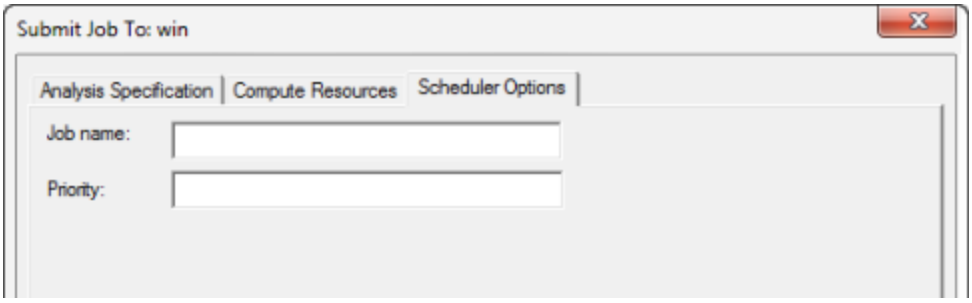


In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, 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.

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 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 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integration with Grid Engine (GE)

The Grid Engine (GE) scheduler (also known as OGE and SGE) is only supported on Linux. With GE, jobs may be submitted in any of the following ways:

- Using GE commands (qsub, etc.) or the SGE gui (qmon)
- 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

Ansys Electromagnetics products support Grid Engine (GE) for Serial analysis, Multi Processing and Distributed Analysis. Models with parametric sweeps can use Large Scale DSO. With GE, the Ansys EM job doesn't require graphics. Ansys EM job's progress can be monitored through GE commands or through Electronics Desktop's Job Management interface.

GE Job Management

You can use Ansys Electronics Desktop to submit batch jobs to GE and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select SGE/OGE/GE as the scheduler.
2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to GE.

   GE-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify the **ParallelEnvironment** parameter. If you do not specify anything, the scheduler will select a parameter.
   - On the **Compute Resources** tab, select **Specify node list** if you wish to specify the nodes. In a computing environment where the available cores are not uniform, you can use this to have control over which resources your job will use. If your Analysis configuration contains a node list, you can use Populate this Page from Analysis Configuration.
   - Memory resource behavior is dependent on the GE version as well as the particular scheduler settings. The output of the `qconf -sc` command shows all of the complexes available to GE schedulers. For a complex to be considered valid for memory resource selection, it must be of type "MEMORY", have relop "<=", be requestable (could be "forced" as well, instead of "YES"), and be consumable. At

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

least one valid memory complex must be available to submit jobs using the Automatic Cores and RAM method.

For Univa GE 8.3 and later, there is an additional column for whether resources are available to a preempting job after preemption of a running job. It is up to the cluster administrator to determine the appropriate Available After Pre-Emption (aapre) setting for memory complexes. This setting is ignored for memory complex validation.

Determining the correct memory complex by default is error-prone. Because the correct choice of memory complex can vary from cluster to cluster, the memory complex selection is now exposed by default under the compute resource selection parameters, allowing the user to make the selection without having to set an environment variable. Only complexes that have been validated (meeting the requirements specified above) can be selected. The cluster administrator (or someone who has knowledge of the specifics of the cluster in use) should be able to determine the correct memory complex to use for Ansys Electronics jobs. To disable exposure of this selection option, the following environment variable can be set to “0”: ANSOFT_SGE_ENABLE_MEM_RES_ATTRIB

- On the **Scheduler Options** tab, you can **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.

Click **Preview Submission** to view the qsub commands to be used to submit the job.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Installation of Ansys EM Tools on GE

Windows:

1. Install on every node of cluster
2. Setup 'temp directory' to a path that is same on all nodes. For example, c:\temp

LINUX:

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 to installation directory

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 desktop scheduler GUI is run the same node as the job submission node, no other configuration is necessary: installation is sufficient. You select the scheduler through the desktop GUI. You need to ensure that scheduler commands are available in the path before you launch desktop.

> **Note:**
>
> There is no need to install RSM unless the 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 nodes is a node in the cluster that can run the Ansys Electromagnetics desktop in graphical mode. A job submission node is a node in the cluster in which job submission commands are available.

Turn OFF firewall between cluster nodes.

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. The case of the SchedulerName string is not significant on Microsoft Windows. In Ansys Electromagnetics Suite 2026 R1, the 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.

SGE Details

For SGE, the ConfigString entry must contain the search path for the SGE commands. It may contain a single directory, the directory containing the SGE commands. Alternatively, it may be a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

path, with directories separated by the colon character ":", where the SGE command directory appears before any other directory containing files with the same name as any SGE commands.

Example ansoftrsmservice.cfg configuration file:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='sge'
'ConfigString'='/opt/sge6.2u4/bin/lx24-amd64'
$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 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: Selecting a Scheduler.

The environment should be configured so that all SGE commands are found using the standard search path. In particular, search for the following commands in the search path should result in the SGE command being found: "qsub", "qdel", "qstat", and "qconf". No other command with the same name should appear before the SGE command in the search path.

GE Commands for Information about Jobs and Cluster Configuration

The following GE commands are especially useful for getting information about the cluster configuration or for getting information about running or completed jobs. This list only contains a few of the most common commands. Consult Grid Engine help for more information.

qconf -help: The first line displays the SGE version

qacct -j *job-id*: Displays a log of the completed job with the specified job ID (if accounting is enabled)

qstat -j *job-id*: Displays a log of the running job with the specified job ID

qconf -sc: Show all complex attributes

qconf -spl: Show a list of all parallel environments

qconf -sp *pe-name*: Show details of the parallel environment with the specified name

qconf -sql: Show a list of all queues

qconf -sq *queue-name*: Show details of the queue with the specified name

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

qconf -sconf: Show configurations

Submitting Ansys EM SGE Batch Jobs via the Command Line

The SGE qsub command may be used to submit Ansys EM jobs. Typical command formats are:

```
qsub qsub_args ansysEM_exe ansys_args
qsub qsub_args job_script
qsub qsub_args[ -]
```

where:

- *qsub_args* are the options of the qsub command,
- *ansysEM_exe* is the pathname of the Ansys EM tool executable to launch,
- *ansys_args* are the arguments to the Ansoft tool command, and
- *job_script* is a shell script containing the Ansys Electromagnetics desktop command to run.

In the first format, the Ansys EM desktop command and its arguments are specified on the **qsub** command line. In the second format, the pathname of a shell script containing the Ansys EM desktop command and its arguments is specified on the **qsub** command line. In the third format, the command is omitted or replaced with a hyphen; this indicates that the command or script will be taken from stdin.

Quoting Ansys EM Command or Arguments for SGE

If the Ansys EM tool executable pathname (*ansysEM_exe*) or any of the arguments of the Ansys EM tool command (*ansysEM_args*) contain characters which are interpreted by the command shell, then these special characters must be properly quoted to ensure that the correct command is launched by SGE. This is especially important when using the first form of the **qsub** command, as the Ansys EM desktop command is processed by the shell twice in this case. It is processed by the shell when the **qsub** command is processed, and again when the job is started.

Serial SGE Batch Jobs

In general, Ansys EM batch jobs may be submitted as SGE serial jobs without any special considerations.

See Monitoring Ansys EM SGE Batch Jobs for options for monitoring Ansys EM batch jobs.

Parallel SGE Batch Jobs

When an Ansys EM batch job is run as an SGE parallel job, the SGE scheduler will select the hosts for the distributed analysis job, and start the desktop process on one of these hosts. The desktop process will obtain the list of hosts from the SGE scheduler, and start analysis processes, as needed, using the SGE scheduler facilities. To run an SGE parallel job, the job must be submitted to an SGE parallel environment (PE).

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 qmaster tcp port is not configured as a service, but rather via the environment variable SGE_QMASTER_PORT, this variable must be set in the Ansys EM batch job environment. This is needed because the ANSOFT EM desktop uses the "qrsh -inherit" command to launch engine processes.

See Monitoring Ansys EM SGE Batch Jobs for options for monitoring Ansoft batch jobs.

Setting Up an SGE Parallel Environment (PE)

To allow Ansys EM batch jobs to distribute analysis engines to multiple hosts, the job must be run in a parallel environment (PE) in which the control_slaves parameter is set to TRUE. This setting is required to allow the Ansys EM desktop to start analysis engines on hosts other than the local host.

Here is a sample parallel environment configuration:

```
pe_name ans_test1
slots 999
user_lists NONE
xuser_lists NONE
start_proc_args /bin/true
stop_proc_args /bin/true
allocation_rule $round_robin
control_slaves TRUE
job_is_first_task FALSE
urgency_slots min
accounting_summary TRUE
```

The user_lists and xuser_lists parameters are ACLs (access control lists) used to control which users have permission to use the parallel environment. The user_lists setting gives permission to use the PE. The xuser_lists setting denies permission to use the parallel environment. The xuser_lists settings override the user_lists settings.

The start_proc_args and stop_proc_args parameters contain the pathname and arguments for the parallel environment startup and shutdown scripts. No startup or shutdown scripts are needed for parallel Ansys Electromagnetics batch jobs. The setting /bin/true may be used as the value for these scripts; this utility does nothing and returns an exit code indicating success (0).

The parallel environment allocation_rule parameter will affect how the analysis engine tasks are distributed across the hosts allocated to the job. The $round_robin setting distributes the tasks across the hosts in a round robin fashion, resulting in the load being relatively evenly distributed over all of the hosts. The $fill_up setting allocates all slots on a host before distributing the tasks to another host; the result is that most hosts are either fully utilized or completely unused. See the sge_pe man page for other settings for this parameter.

The control_slaves parameter must be set to TRUE, 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.

The job_is_first_task parameter also affects how tasks are allocated. When submitting a job to run in a parallel environment, the number of parallel tasks, n, is specified on the command line. If this setting is TRUE, then the job process is considered one of the tasks, and only (n-1) additional tasks are allocated to the job. If the setting is FALSE, then the job process is not considered to be one of the tasks, and n additional tasks are allocated for the job.

See the sge_pe man page for more information about these and other PE parameters.

A parallel environment does not run tasks directly. Instead, the tasks are distributed to queues associated with the parallel environment. In order to complete the setup of a parallel environment, one or more queues need to be associated with the parallel environment. The queue pe_list parameter is used to specify the parallel environments (PEs) supported by the queue. This is an important step; **if no queues support a given PE, then jobs submitted to that PE will not run**.

Parallel Batch Job Command Line Considerations

The number of engines run on a host will depend on the total number of distributed engines, and the number of hosts allocated to the job. The memory required on a host depends on the number of engines running on the host and on the memory needed for each engine. The **qsub** command **-l resource=** *value*,... or **-q** *queue_list* > command line options specify that the parallel batch job run on machines with sufficient memory and other resources.

Monitoring GE Serial and Parallel Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for GE serial jobs and GE parallel jobs.

GE qstat Command

The SGE **qstat** command may be used to display information on jobs and queues. If the **-j** option is included, then information on jobs is displayed. If the option includes a job list (**-j [** *job_list* **]**), then the displayed information is limited to the jobs in the job list.

The **-u** *user*,... option limits the output to jobs associated with users in the user list. If the **-u***user*,... option is not specified, then information on queues or jobs of the current user are displayed.

The **-t** option displays extended information about the subtasks of each displayed job. This is equivalent to the **-g t** option. The **-r** option displays extended information about the resource requirements of the displayed jobs.

See the SGE manual pages 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.

Ansys EM Desktop -monitor Command Line Option for SGE

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.

The SGE scheduler redirects the standard output and standard error streams of batch jobs to files specified in the **qsub -o [[** *hostname* **]:]** *path*,... and the **-e [[** *hostname* **]:]** *path*,... command line options, respectively. If either option is not specified, then the associated stream is redirected to the default file pathname.

The **qsub -j y[es] | n[o]** controls whether the standard output and standard error streams are merged. If the y or yes value is specified, then the standard error stream is merged into the standard output stream. If the **-e** *host_and_path* option is also specified in this case, the *host_and_path* setting is ignored. If the n or no value is specified, or if this option is not specified, then the standard error stream and standard output stream are not merged.

You 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.

Example SGE qsub Command Lines

All of the following examples show how to submit Linux HFSS jobs on SGE, but similar command lines will work for all Ansys Electromagnetics products.

Serial job using command line:

```
qsub -b y /opt/ansys_inc/v261/AnsysEM/ansysedt -ng -BatchSolve
 ~/projects/OptimTee.aedt
```

- The **-b y** option indicates that hfss is launched directly from the command line, instead of using a script.
- No queue is specified, so the default queue will be used.

Serial job with a hard runtime limit of 15 minutes:

```
qsub -b y -l h_rt=00:15:00 /opt/ansys_inc/v261/AnsysEM/ansysedt
 -ng -BatchSolve ~/projects/OptimTee.aedt
```

- The -l h_rt=00:15:00 option indicates that this job has a "hard" runtime limit of 15 minutes.

Serial job using a script, with a runtime limit specified in the script:

```
qsub ~/sge/scripts/OptimTee.csh
```

- The **-b y** option is absent, so the script ~/sge/scripts/OptimTee.csh will be run when the job starts.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The script file OptimTee.csh may contain SGE directives in addition to the command(s) to run. In this example, a directive with a hard runtime limit if 15 minutes is included in the script.

Script file contents:

```
#!/bin/csh
#$ -l h_rt=00:15:00
/opt/ansys_inc/v261/AnsysEM/ansysedt -ng -BatchSolve
 ~/projects/OptimTee.aedt
```

- The SGE directive #$ -l h_rt=00:15:00 is equivalent to including -l h_rt=00:15:00 on the qsub command line.

Distributed processing job using 4 engines:

```
qsub -b y -pe pe1 4 /opt/ansys_inc/v261/AnsysEM/ansysedt
 -ng -BatchSolve -Distributed -machinelist num=4
~/projects/OptimTee.aedt
```

- The **-b y** option indicates that hfss is launched directly from the command line, instead of using a script.
- The **-pe pe1 4** command_line option indicates that this is a parallel job running under the pe1 parallel environment, and that 4 cores or processors are allocated to this parallel job.
- The "-machinelist num=n" option is now required for batch jobs.
- 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, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Issue with qrsh (SGE)

Ansys EM parallel batch jobs use the SGE **qrsh** command to launch engine processes on remote hosts. If the **qrsh** command is not working correctly, then the parallel job is unable to launch engine processes on remote hosts. If this problem occurs, the batch log for the job typically includes one or more error messages indicating that a COM engine was unable to be started on a remote host. If this occurs, the user or cluster administrator should verify that the SGE qrsh command is working correctly, and correct the problem if the SGE **qrsh** command is not working correctly.

The **qrsh** command may be tested by running a simple command on a specified host, such as **qrsh -l hostname=host1 hostname** or **qrsh -l hostname=host1 ls /tmp**, where host1 is the remote host name. The first test should simply echo back the hostname of the remote machine. The second test should list the contents of the /tmp directory 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.

The failures of the SGE qrsh command are associated with the following global sge configuration parameters, listed below with values that may cause the failures:

```
qrsh_command /usr/bin/ssh -t
rsh_command /usr/bin/ssh -t
rlogin_command /usr/bin/ssh -t
```

If these parameter settings are removed, then the SGE built-in mechanisms are used for qrsh, rsh, and rlogin. No problems with the built-in versions have been reported. The SGE qconf -sconf global command may be used to view these parameter settings. The SGE qconf -mconf global command may be used to modify or remove these parameter settings.

Issue with MainWin Core Services for SGE

By default, SGE creates a temporary directory for each SGE batch job, and deletes this temporary directory and its contents when the job finishes. SGE sets the TMP and TMPDIR environment variables of the job environment to point to this temporary directory. Ansys EM desktop software starts the MainWin Core Services on startup, if they are not already running. After the Ansys EM desktop software finishes, the MainWin Core Services time out and automatically shut down. The MainWin Core Services use the TMP and/or TMPDIR directories to store temporary data. If this temporary data is removed before the services shut down, then the services do not shut down automatically. Normally, SGE will remove the temporary directory and its contents before the services time out. The result is that these extraneous service processes run forever. If this problem occurs, each Ansoft batch job starts an additional set of these services that never shut down. This can result in an excessive number of processes running on the host where the Ansys EM desktop is started. The names of the service processes are:

- watchdog
- regss
- mwrpcss

Workaround for Issue with MainWin Core Services

One way to avoid this problem is to modify the environment in which the Ansys EM desktop runs so that the TMP and TMPDIR environment variables do not point to the directory which will be immediately removed by SGE when the job finishes. This can be done by copying the value of the TMPDIR environment variable to the ANS_SGE_TMPDIR environment variable, and unsetting the TMPDIR and TMP environment variables. The services ignore the ANS_SGE_ TMPDIR environment variable, but if this variable is set, then it will be used as the temporary directory for the rest of the Ansys EM software.

Here is an example bash wrapper script that may be used to work around this issue. In this example, the product is hfss, but the same approach will work for any Ansys EM product. In this example, the script is named sge_hfss and is in the AnsysEM software installation directory.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When an Ansys Electromagnetics desktop job is submitted to the SGE scheduler, the script (sge_hfss, in this example) should be submitted instead of hfss. The script will modify the environment, as needed, then start hfss. When the analysis finishes, the script returns the exit status of hfss.

An alternative is to place the script in an arbitrary directory, and modify the script to include an absolute path to the product (hfss in this example).

Script contents:

```
#! /bin/bash

# This script will not correctly process arguments containing
# spaces or other characters special to the shell.

# Create hfss command line
# In this example, sge_hfss and hfss are in the same directory
# An alternative is to use an absolute path for the hfss command
cmd0=$0
cmd="${cmd0/%sge_hfss/hfss} $@"

# Fix environment variables
export ANS_SGE_TMPDIR=${TMPDIR}
unset TMPDIR
unset TMP

# Run the hfss command and return the exit status
${cmd}
exit $?
```

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

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 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:

   - 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, a limit number of tasks per node, and RAM limit as percent.
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:

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 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
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 Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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.

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, 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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
```

- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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>
```

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?

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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:

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 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.

  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

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 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.
- 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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.

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" on the facing page
- "Distributed Analysis and Multiprocessing in the Same Job" on the facing page

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 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
```

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
```

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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:

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 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 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.

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 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

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

```
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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

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" 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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):

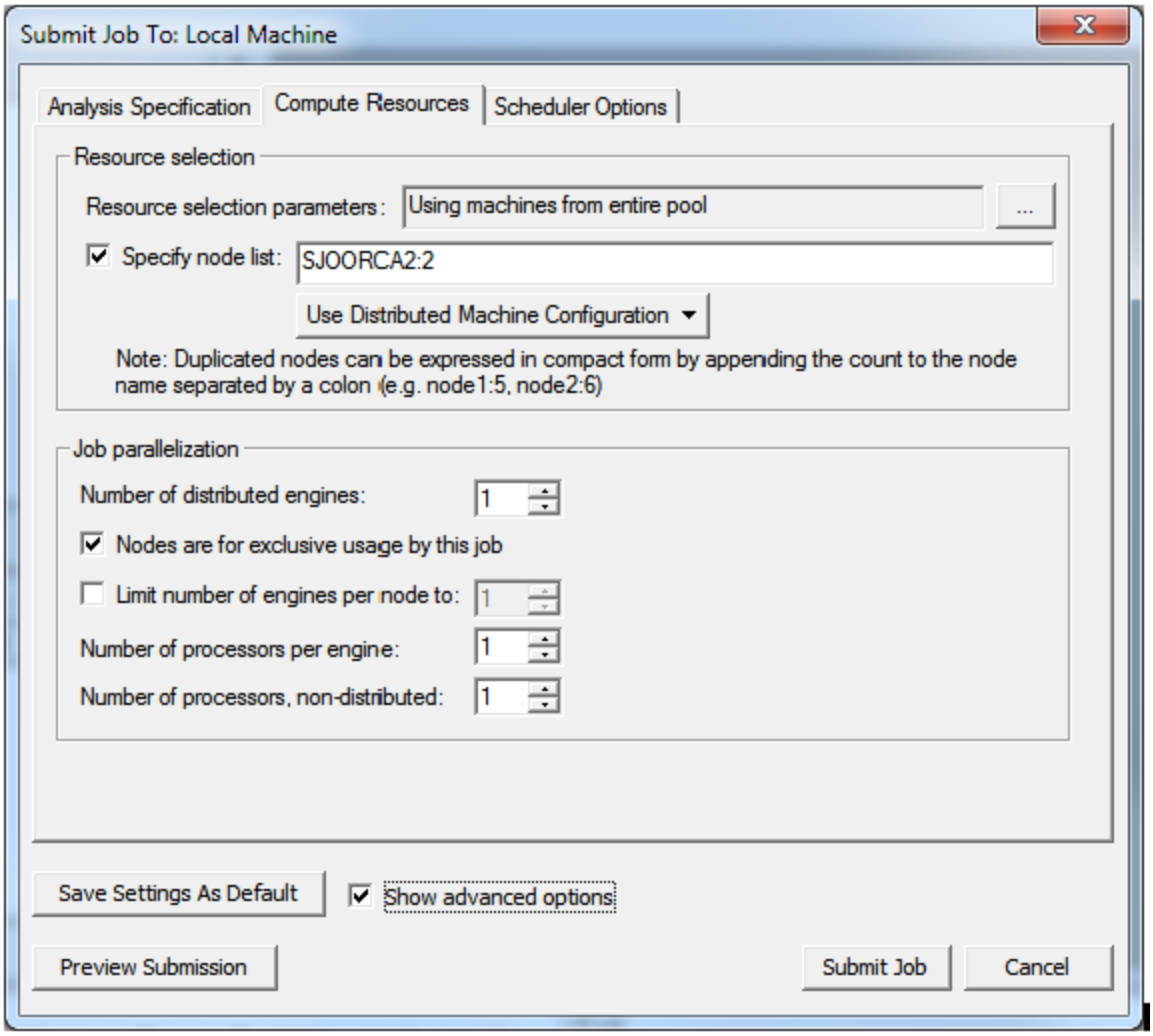
This function returns some basic information about the scheduler with which the scheduler proxy library interacts.

- 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.

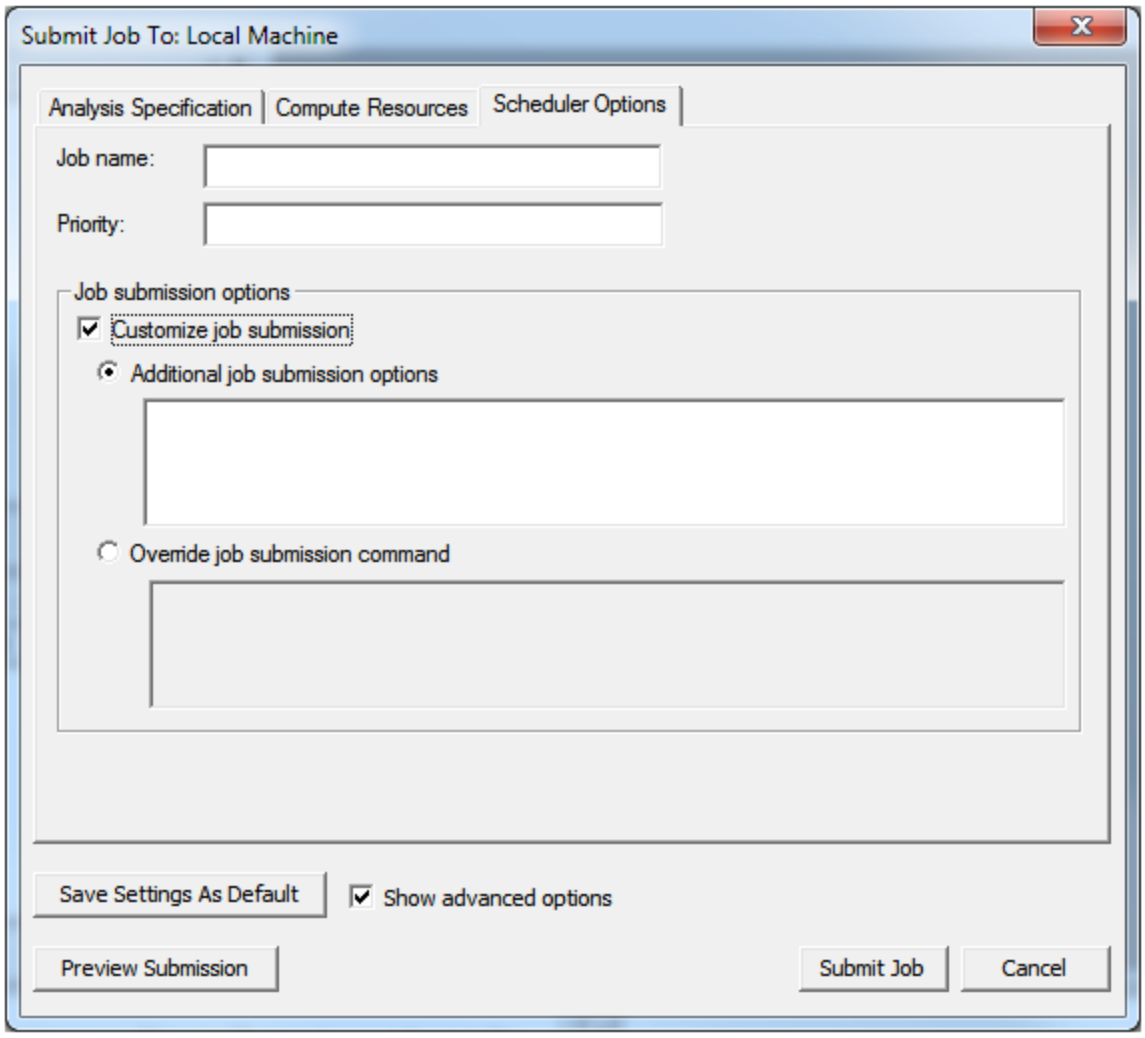
The **computeResources** argument is a reference to an object of type **UniformComputeResourcesStruct**. This **struct** contains zero or more resource attribute settings for the job. Each resource attribute setting consists or a resource name and a resource value. The resource name is the name of one 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Troubleshooting GetThisJobID

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Troubleshooting LaunchProcess

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Installation Details

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Each of the functions to be implemented in the SamplePluginExtension class is described below.

- 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
GetDescription(self)
```

Arguments (excluding self)

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
GetSchedulerDisplayName(self)
```

Arguments (excluding self)

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.

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 be used to launch engine processes for this scheduler, then this function shall return True.

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.

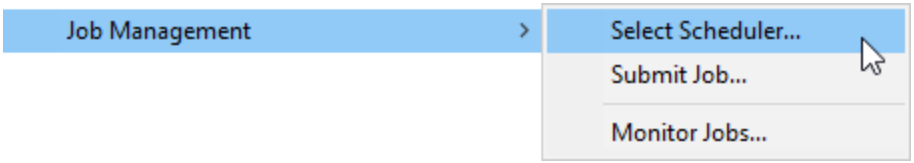
Note:

- 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

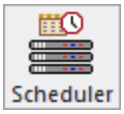
Access the **Select Scheduler** window one of three ways:

- Click **Tools** > **Job Management** > **Select Scheduler...**



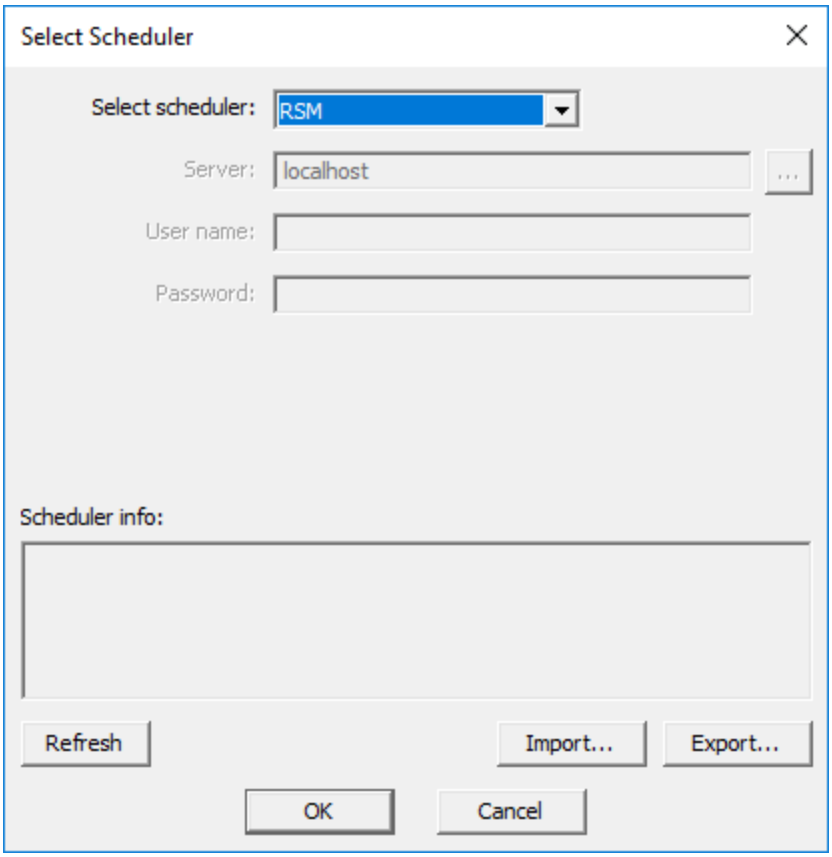
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

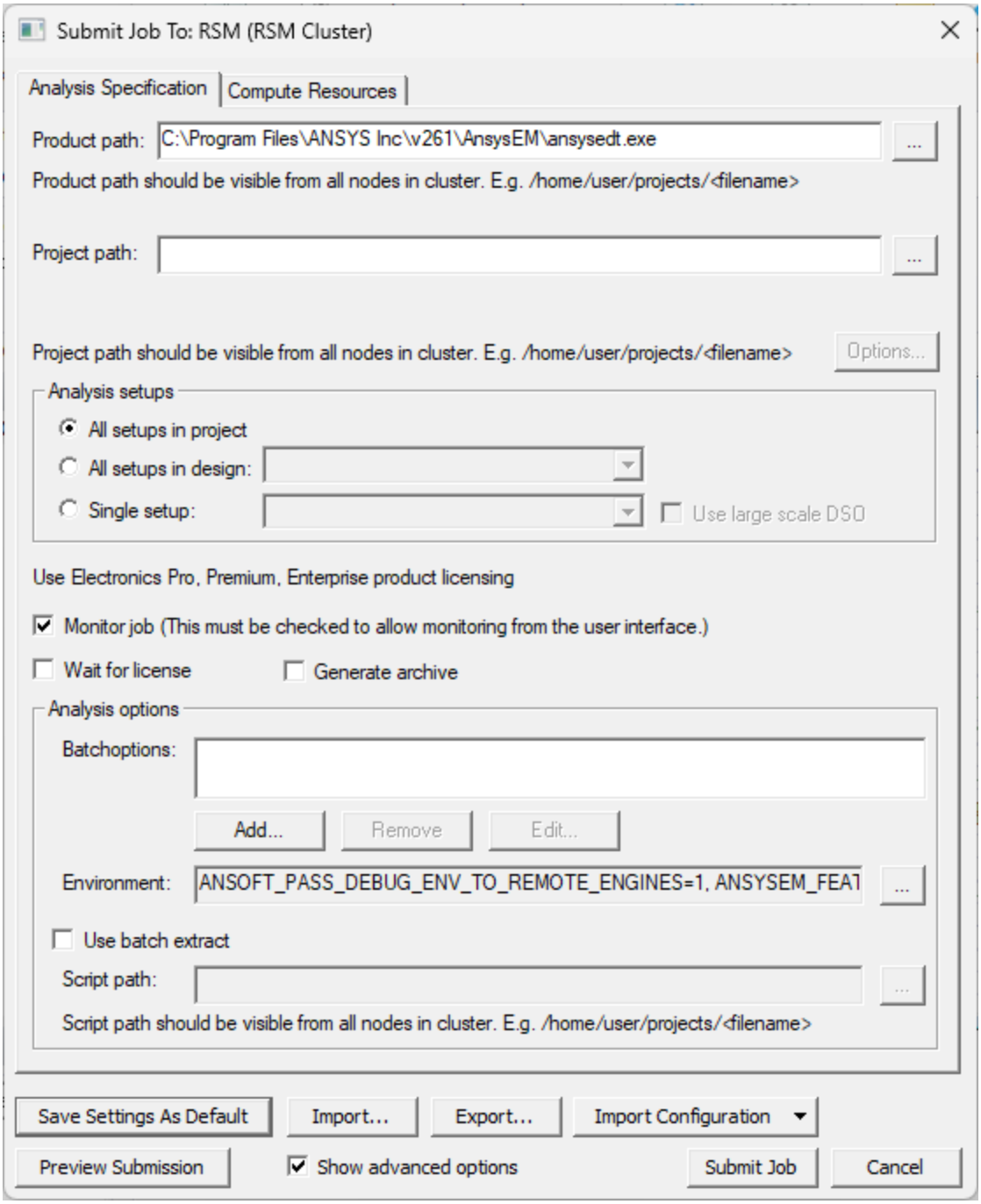
Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn't set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to 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 **HFSS 3D Layout** > **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.

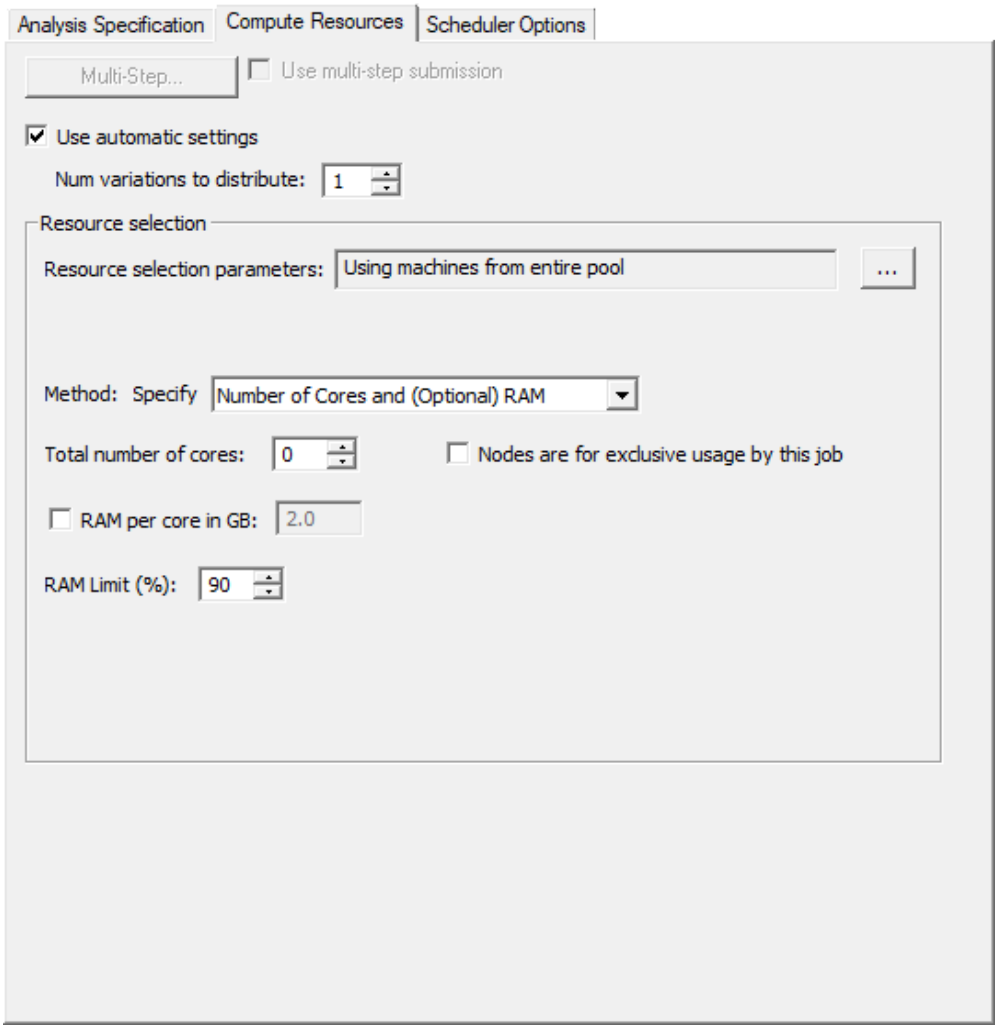
   In the **Analysis Setups** area, select the radio button for **All setups in project**, **All setups in design**, or a **single setup**.

   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.

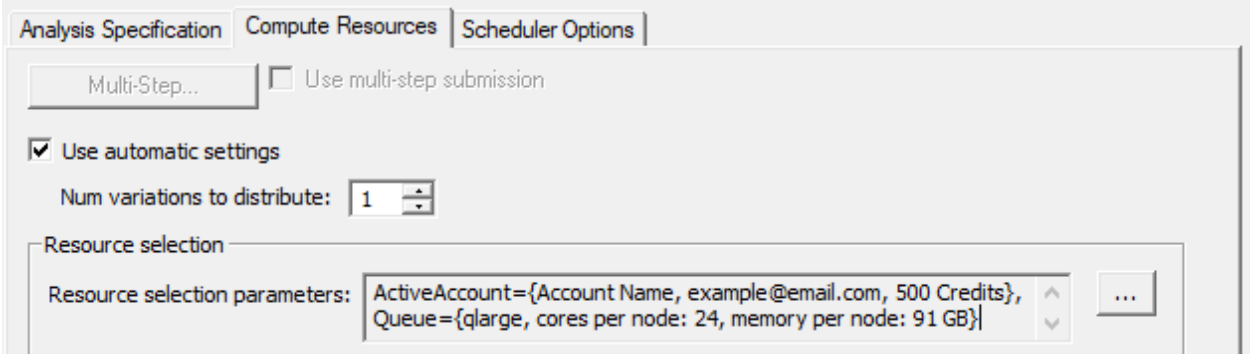
5. If applicable, determine whether or not to **Use multi-step submission**.
6. Determine whether or not to **Use automatic settings**.

> **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:

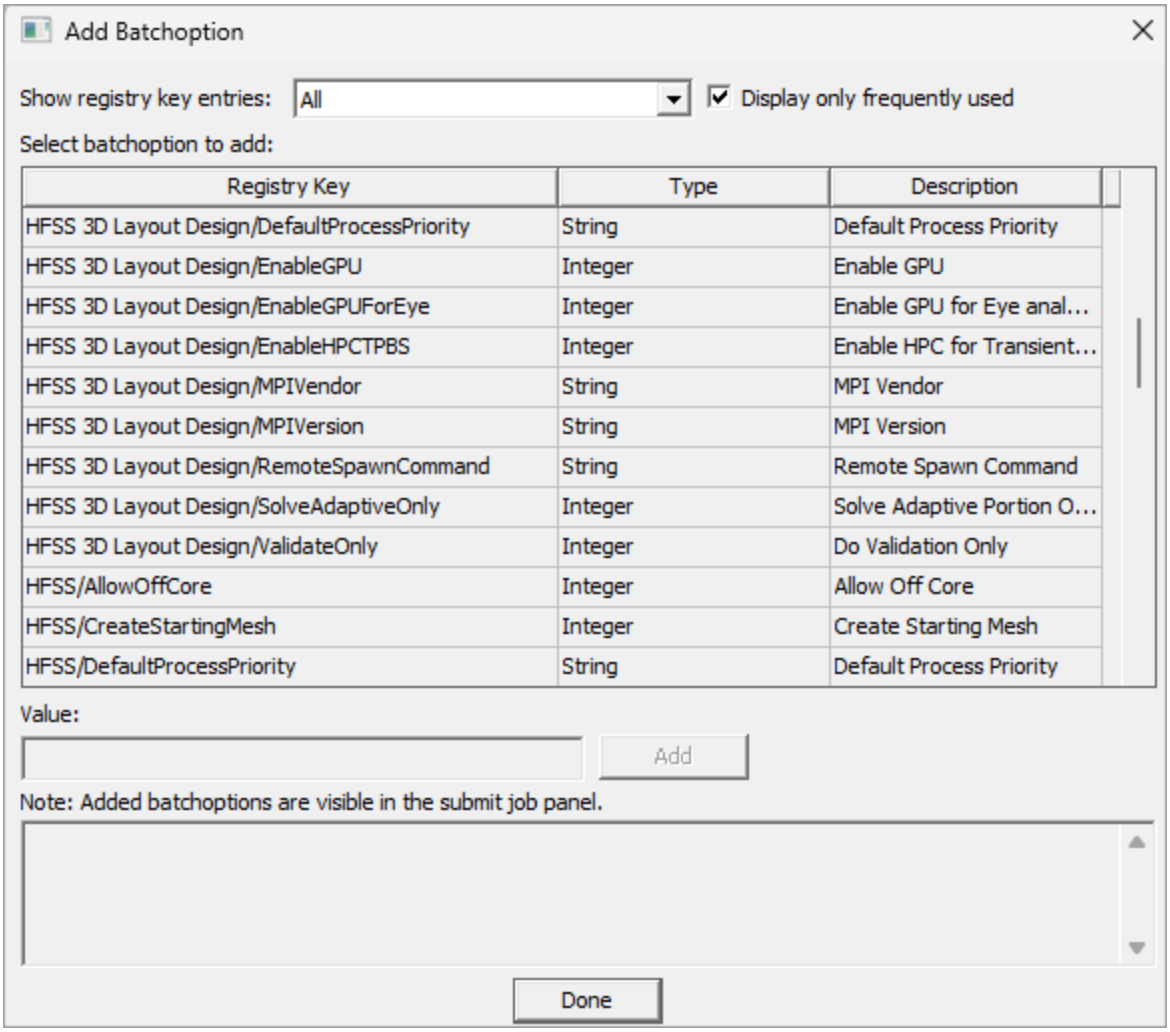
- Batchoptions
- Environment Variables

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.

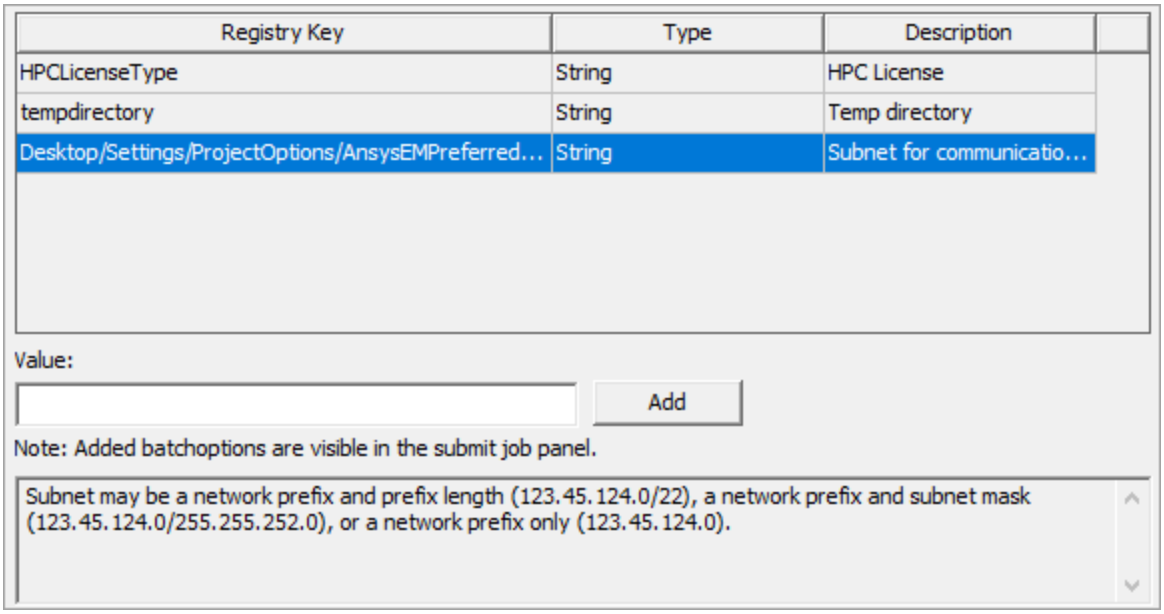


The **Show registry key entries** drop-down menu allows you to select categories of registry keys to display.

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:

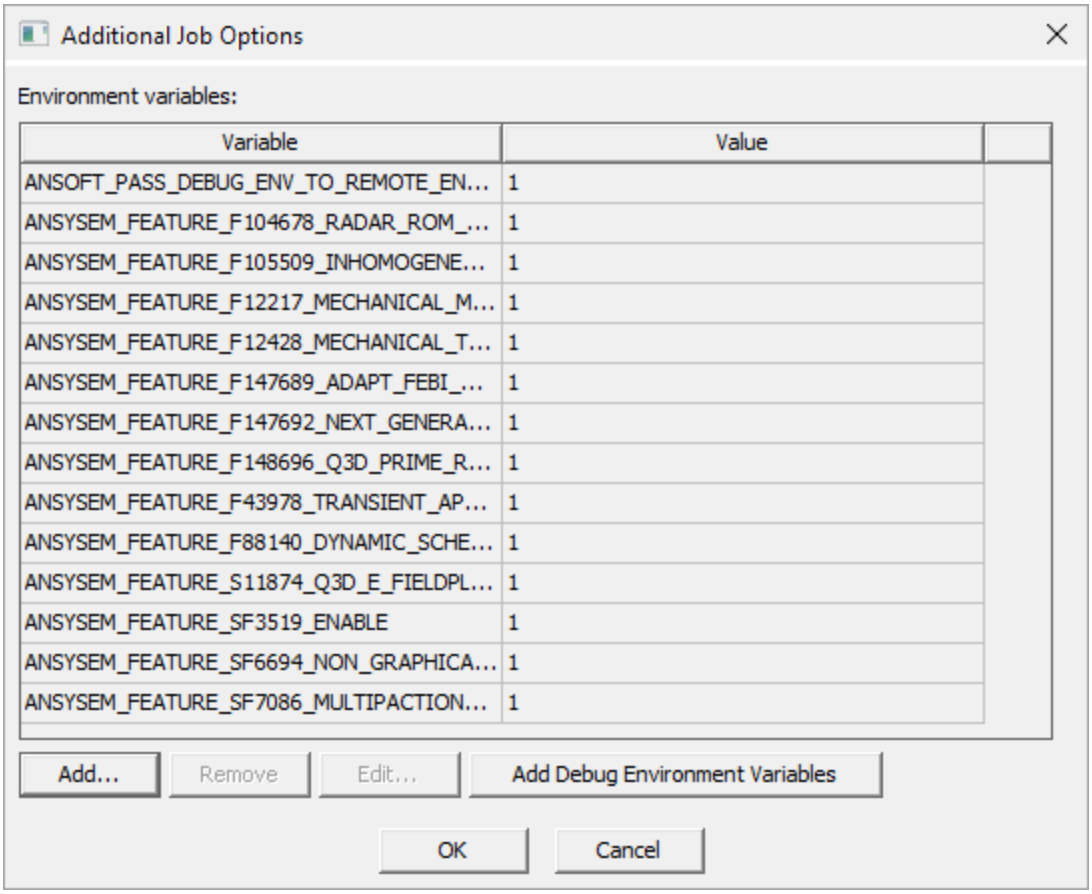
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 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
```


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

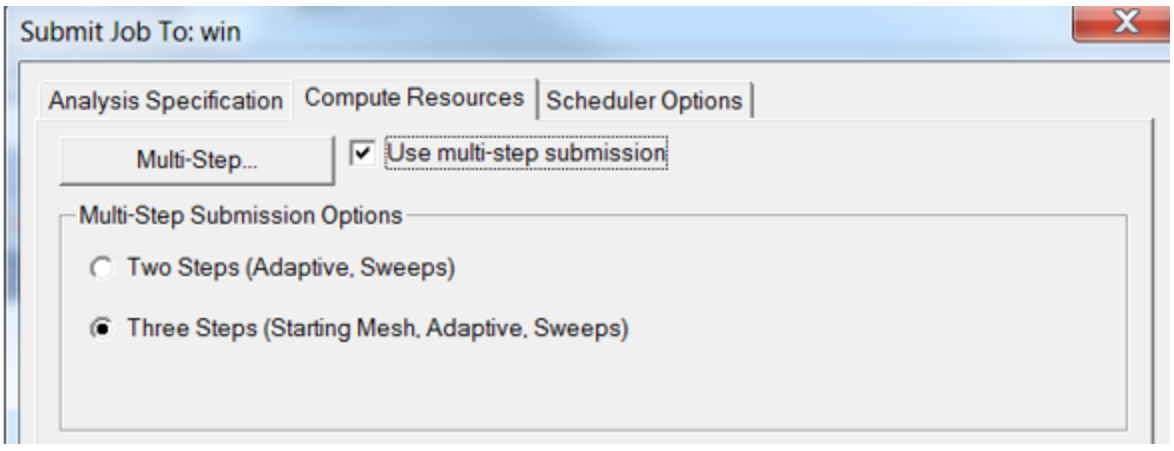
Multi-step job submission allows you to divide the simulation of a project on a cluster as two or more jobs, each of which has unique resource specifications. For example, creating the initial mesh and doing adaptive refinement can use a single machine, while frequency sweeps can easily be distributed 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.

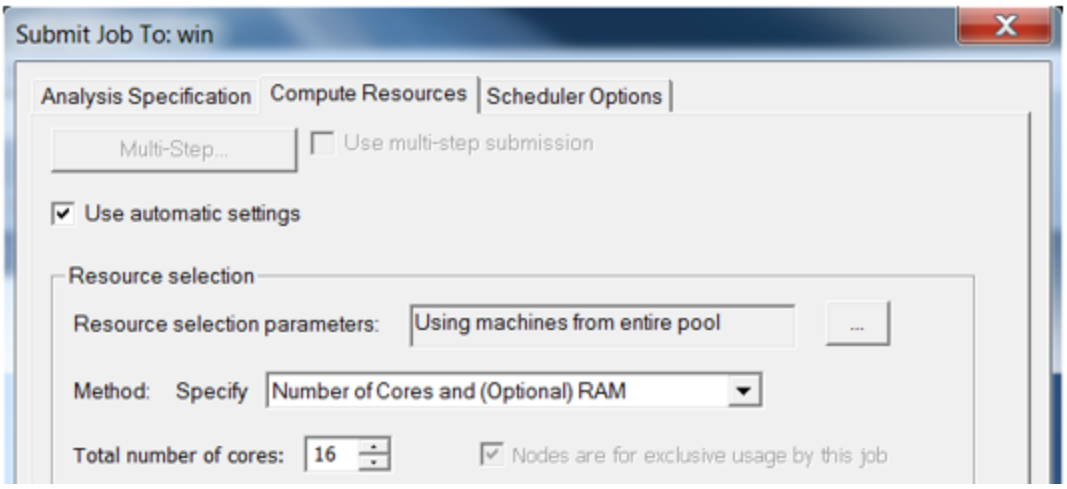
How-to Specify Multi-Step Job Submission

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.

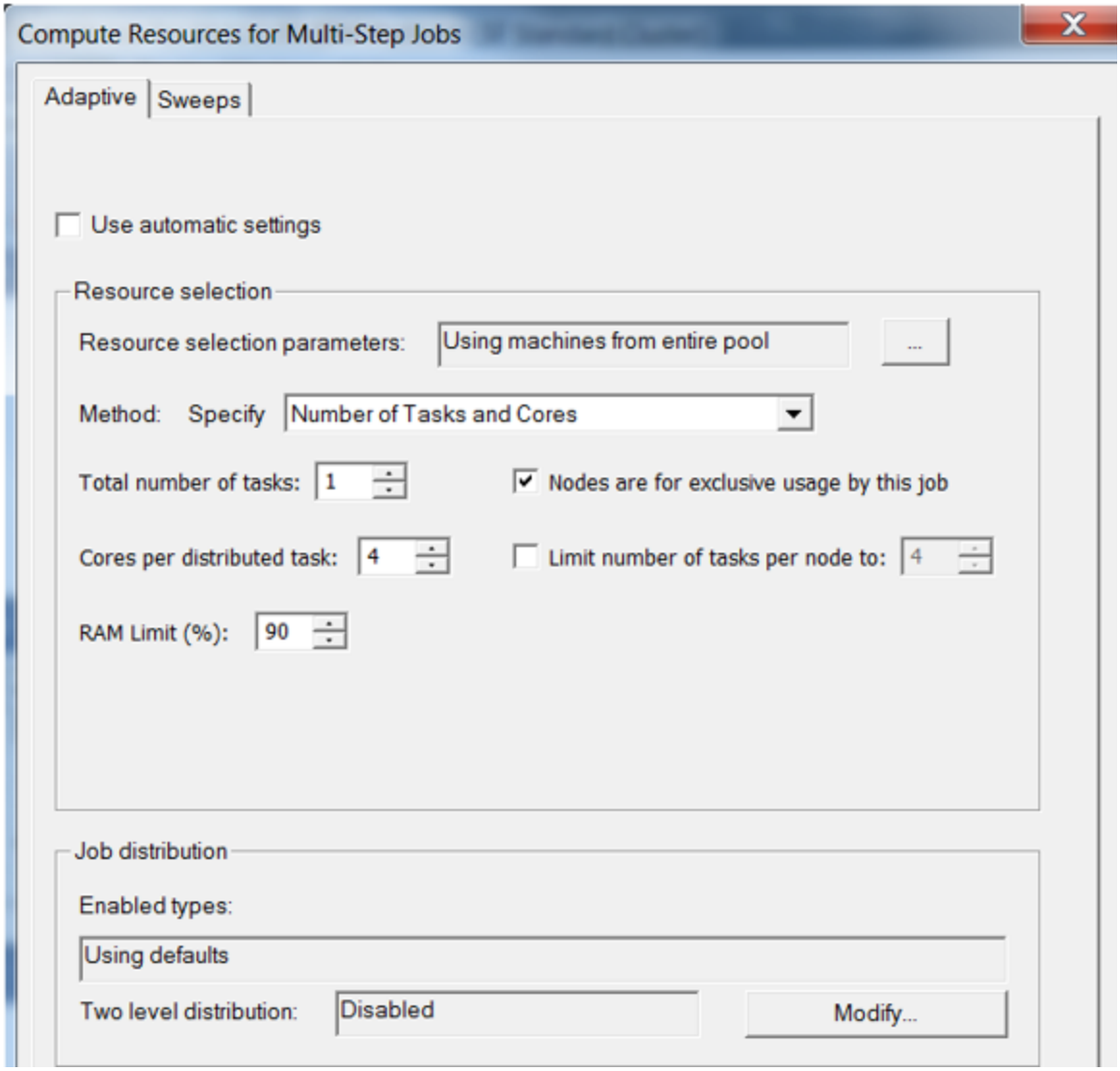


On the **Compute Resources** tab, check the **Use multi-step submission** box and select the appropriate submission option:

- Two Steps for Adaptive and Sweeps
- Three Steps for Mesh, Adaptive, and Sweeps

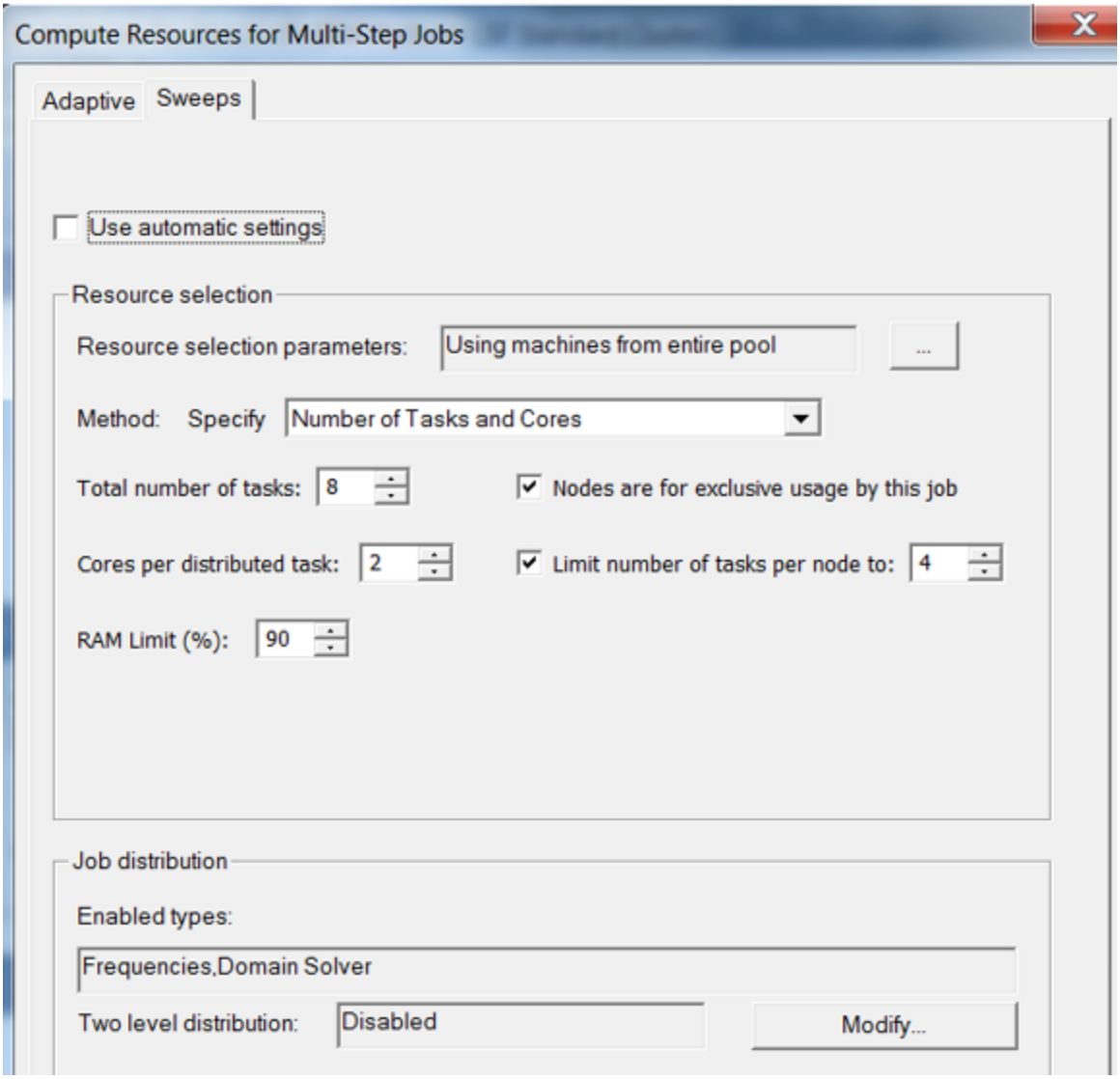
Then click **Multi-Step…**, which will bring up a **Compute Resources for Multi-Step Jobs** dialog box where there is a tab for each step used, that is for potential for Mesh, Adaptive, and Sweeps.

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.

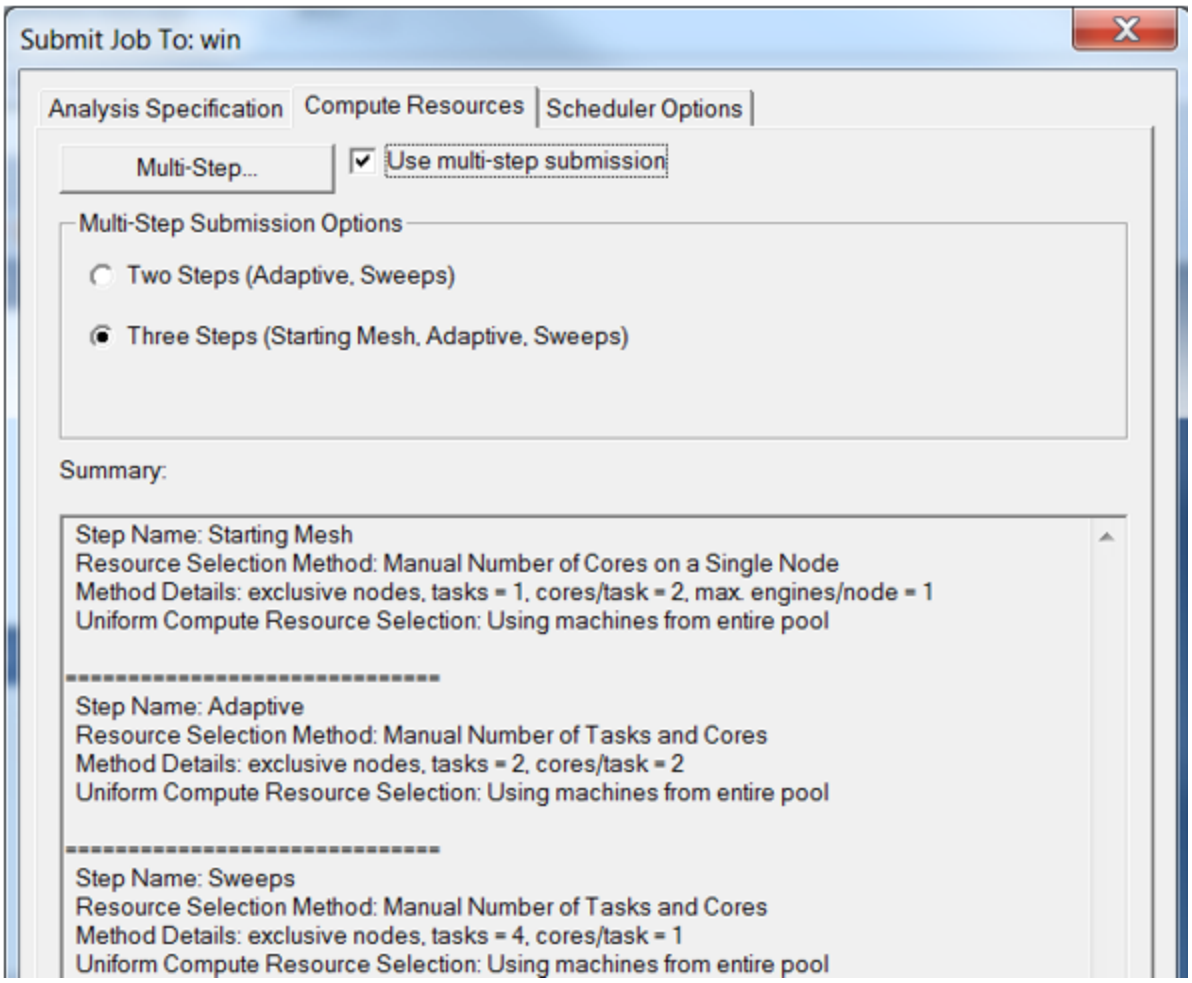
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



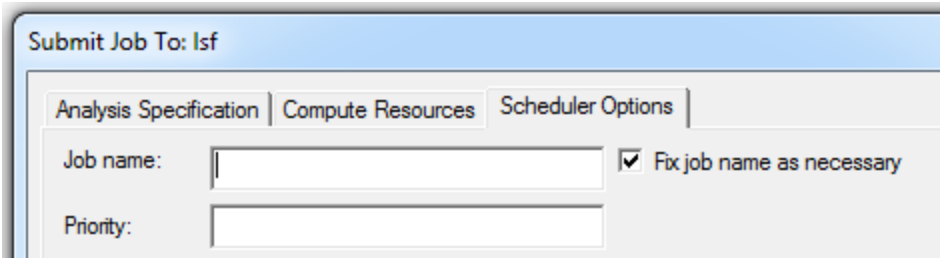
Set the desired compute resources for each step and click **OK**.

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.



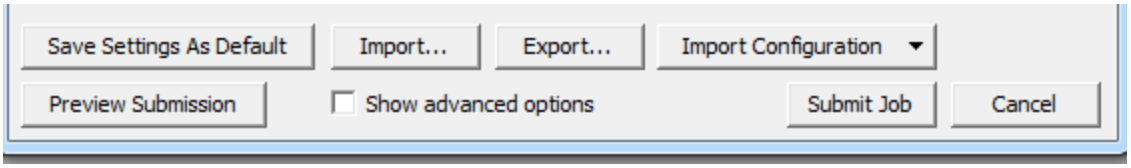
If desired, set the job name on the **Scheduler Options** tab. When the jobs are submitted, each will have a name given by job name (if any) appended with the name of the step for the job. For example, if the job name entered is “MultiStep”, then the individual jobs might be named “MultiStep-Adaptive” 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

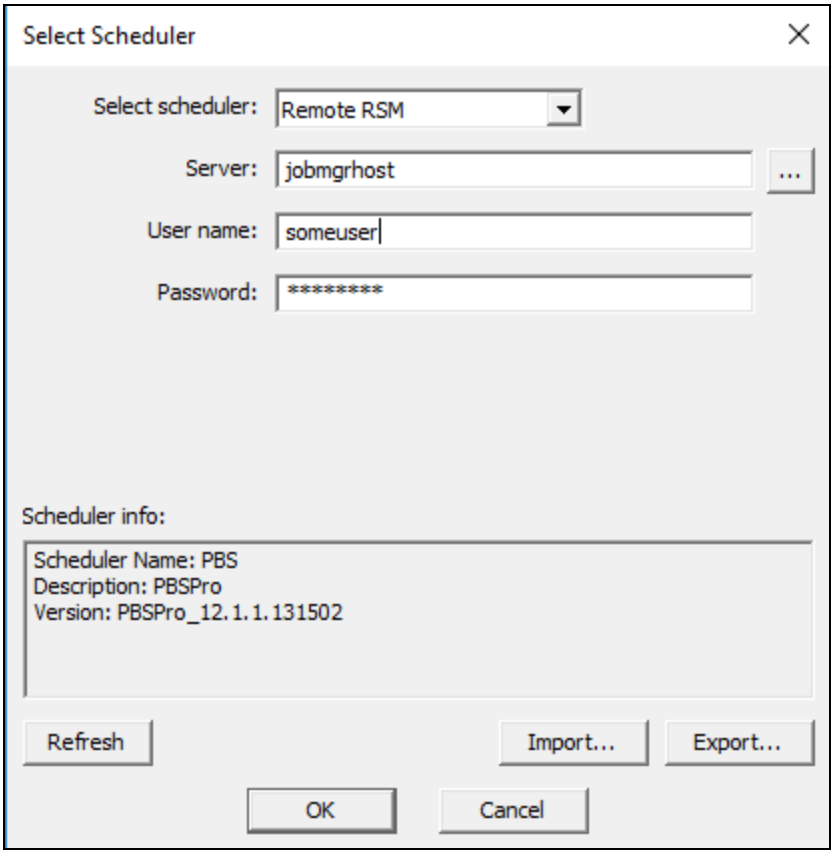
This feature may be used with all Linux schedulers for which job submission from the GUI is supported:

- 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.

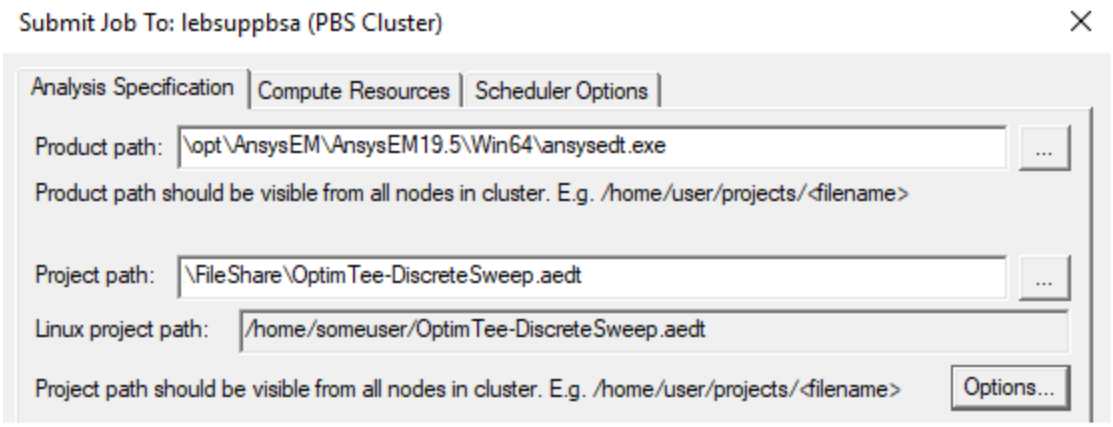


Submit Job

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.

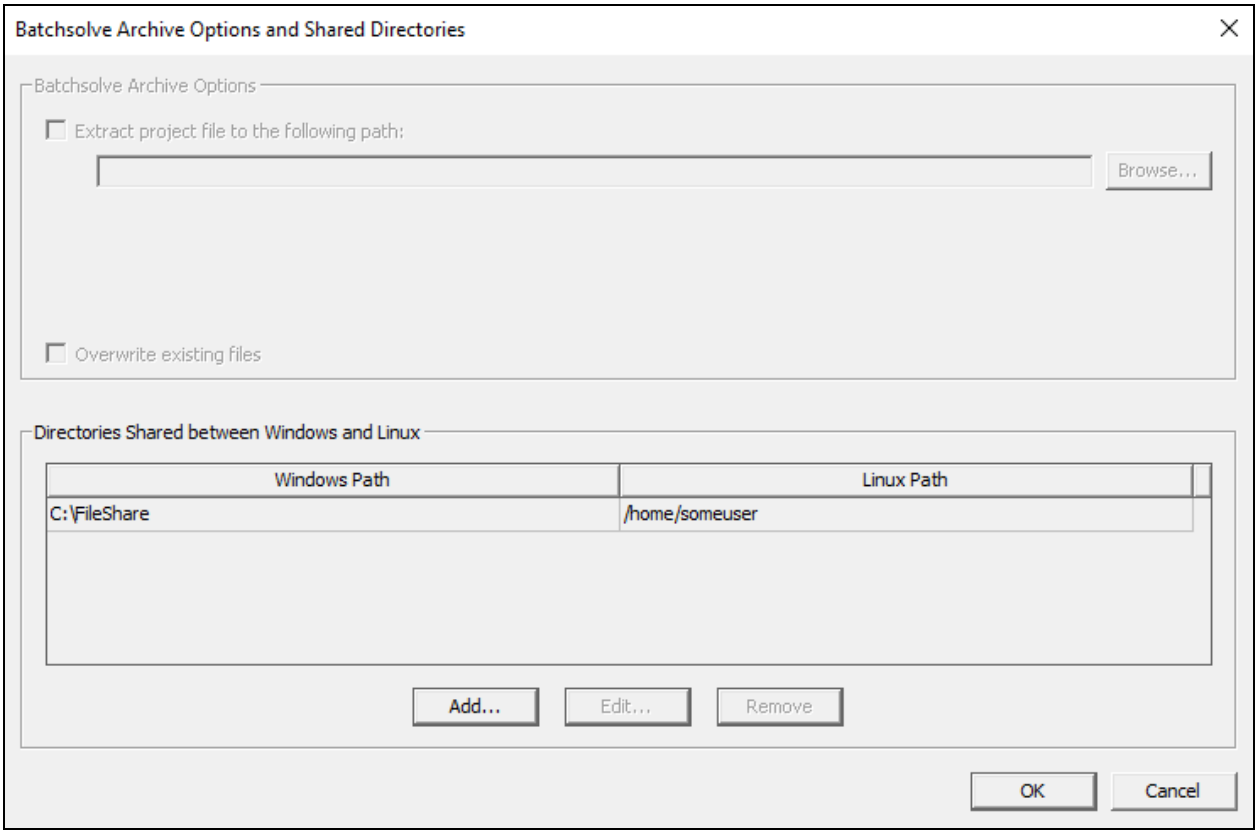
You must enter the Windows path of the project file in the Product path edit control or use the browse button (labeled "...") to select the Windows path of the project file. If the Linux path of the project file can be determined from the specified Windows path and the directories shared between Windows 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.



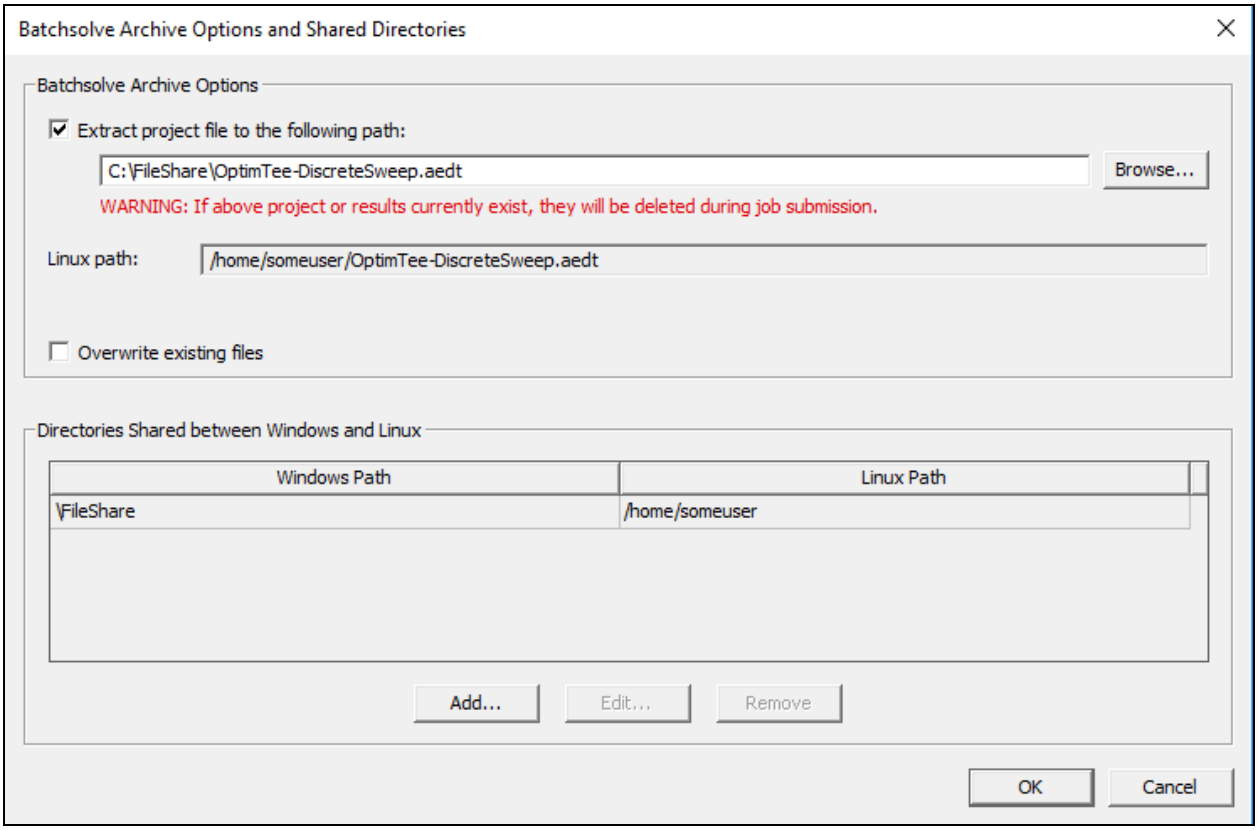
The **Options** button opens a window that you can use to specify archive options for a job. The lower portion of this window allows you to specify one or more directories shared between Windows and Linux. The mapping of directories between Windows and Linux is shown in a grid which displays the Windows 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.



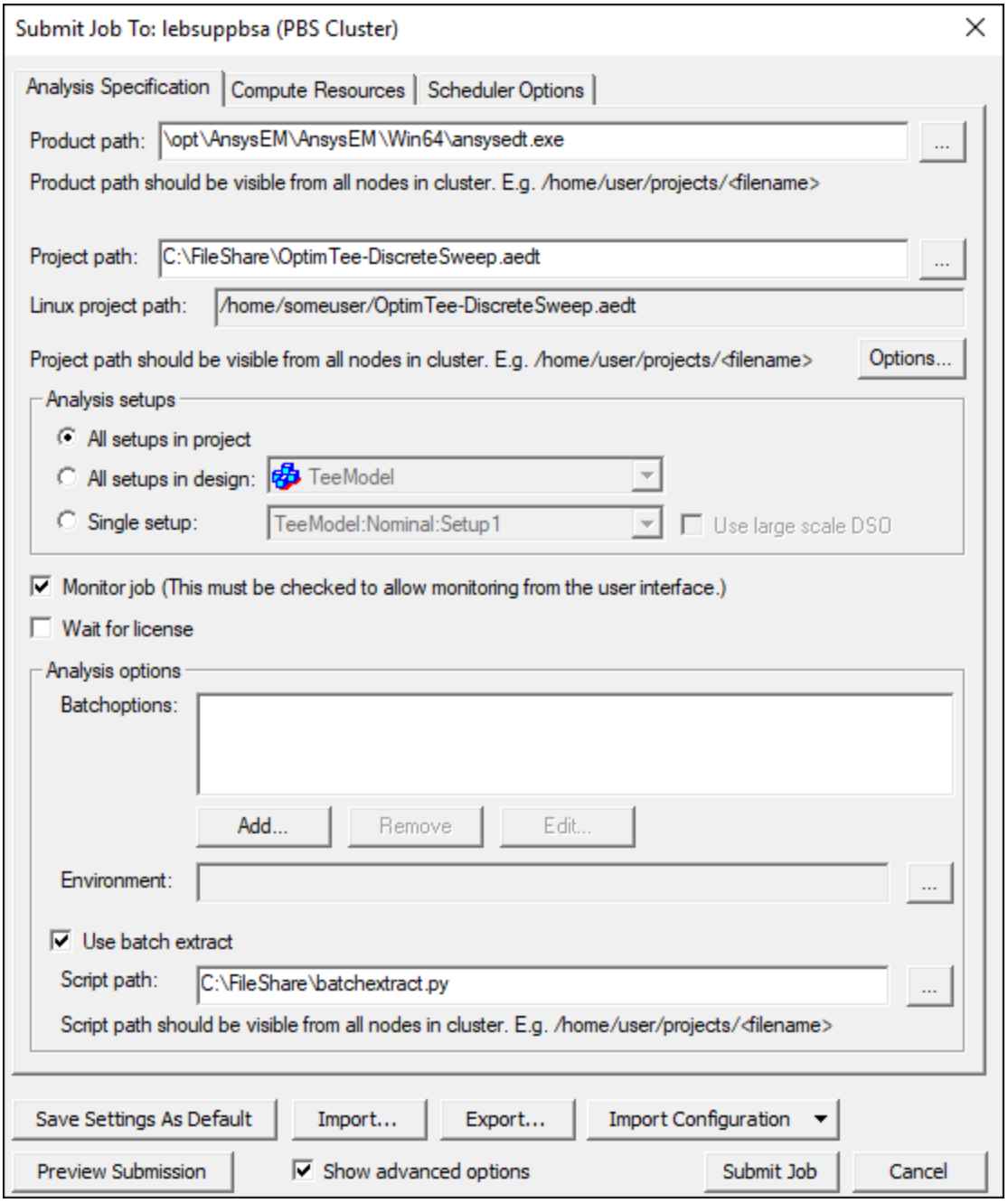
If you specify a project in an archive, the window may be used to specify the Windows pathname of the project to be extracted from the archive. If this is done, the Linux pathname of the target project is determined from the directories shared between Windows and Linux and shown in the upper portion 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** > **HFSS 3D LayoutScripting 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.

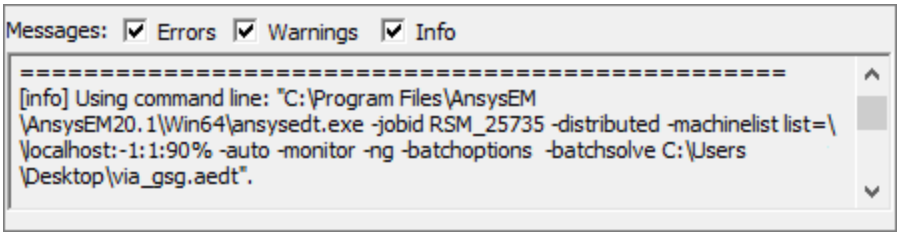
The **Monitor Job** window contains the following areas:

- **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.
- **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

- **Messages** – Displays errors, warnings, and job information. Use the check boxes to choose which of these to display.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Progress** – Displays progress bars when tasks are currently in progress.

The arrow button opens a menu that allows you to either **Abort** that analysis, or perform a **Clean Stop**.

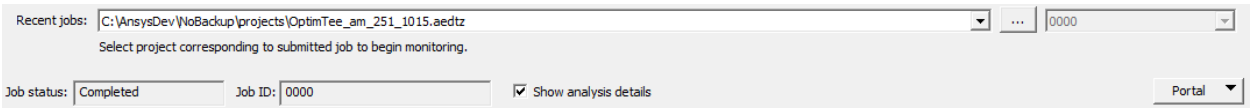
- **Refresh Job Output** – Allows you to select either manual or automatic refresh for the **Monitor Job** window.

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.

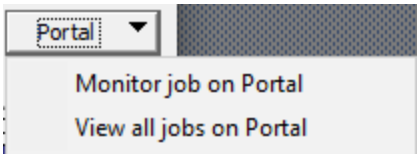
Monitoring Ansys Cloud Burst Jobs

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**.



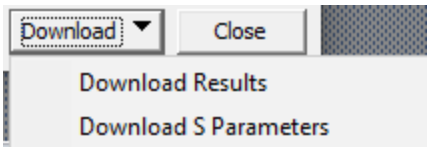
The **Portal** drop-down button provides a link to the Ansys Portal, where job details and additional monitoring information are available. For more information, see the Ansys Portal Guide.

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 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.

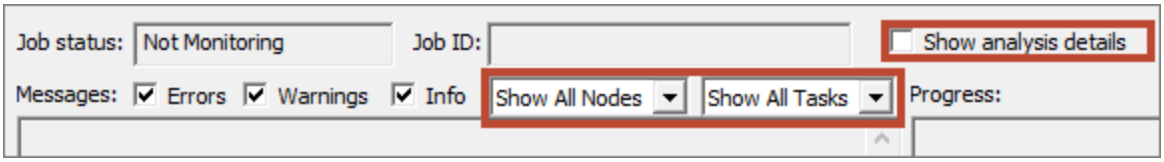


Monitoring Large Scale DSO Jobs

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.



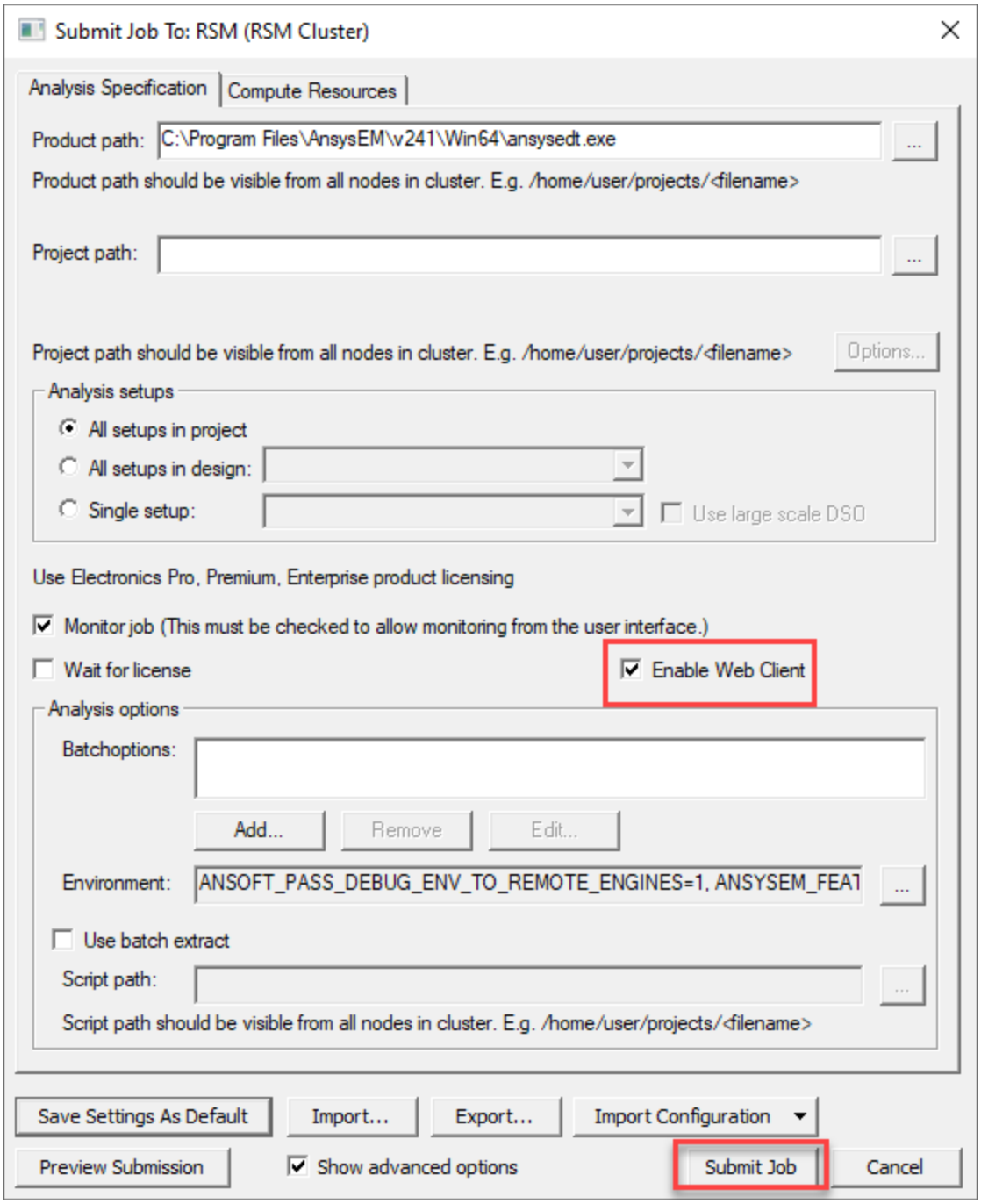
To leave this mode, select **Show analysis details**.

Web Client for Batch Solve Monitoring and Reporting

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

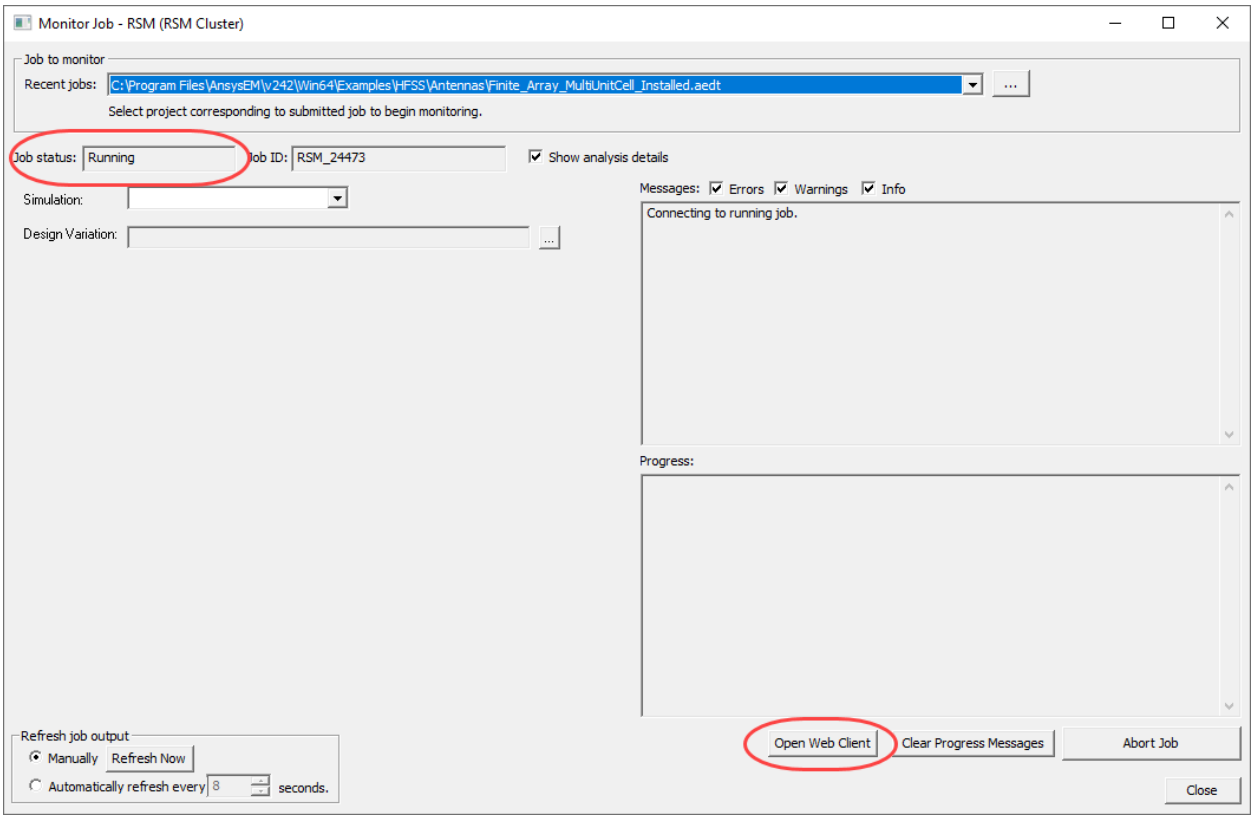
There is a Beta Option to use a Web client for batch solve monitoring and reporting. If you set the Beta Option under **Tools>Options>General Options> Desktop Configuration** for "Enable Web Client for Monitoring," you will have access to the feature.

After you have selected the scheduler to use, when you access the **Submit Job To:** dialog, you well see Enable Web Client checked.



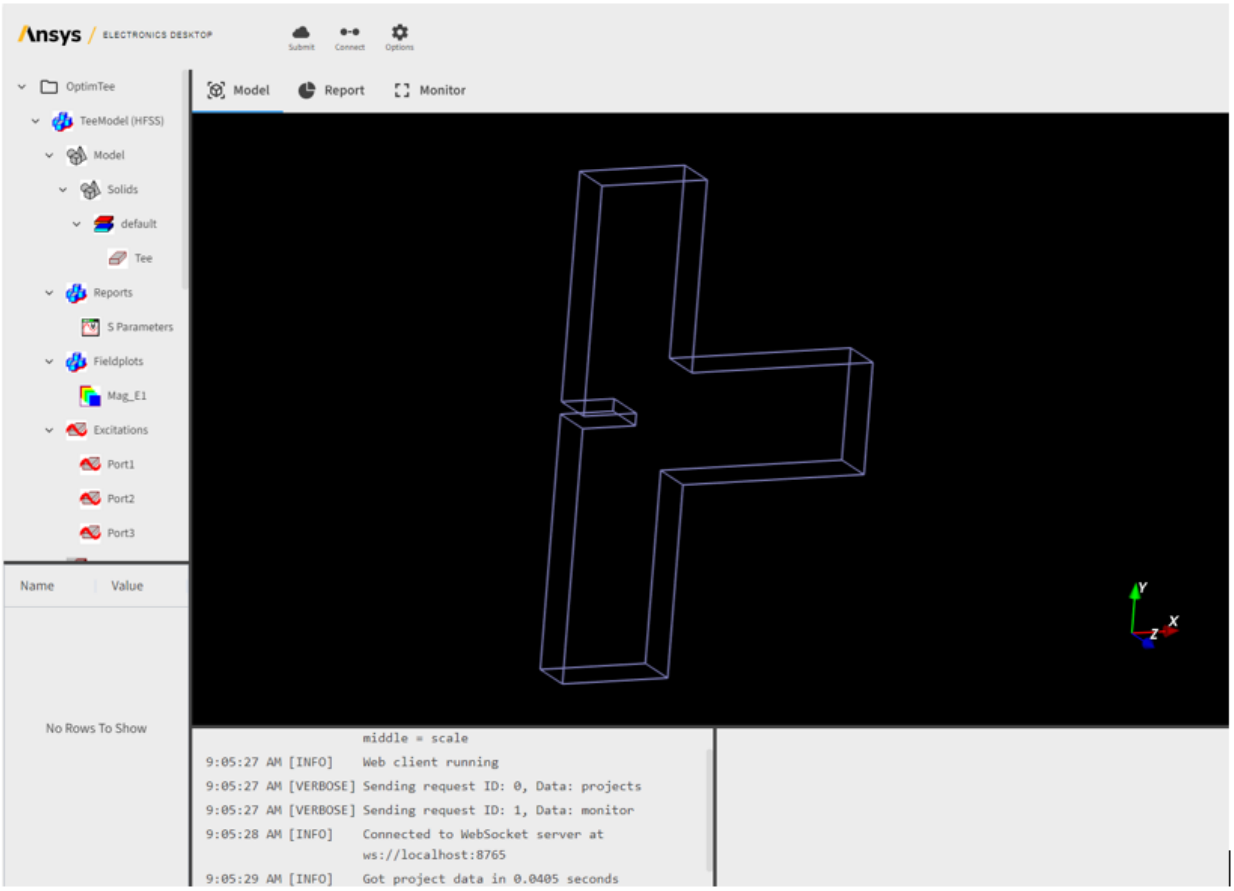
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.



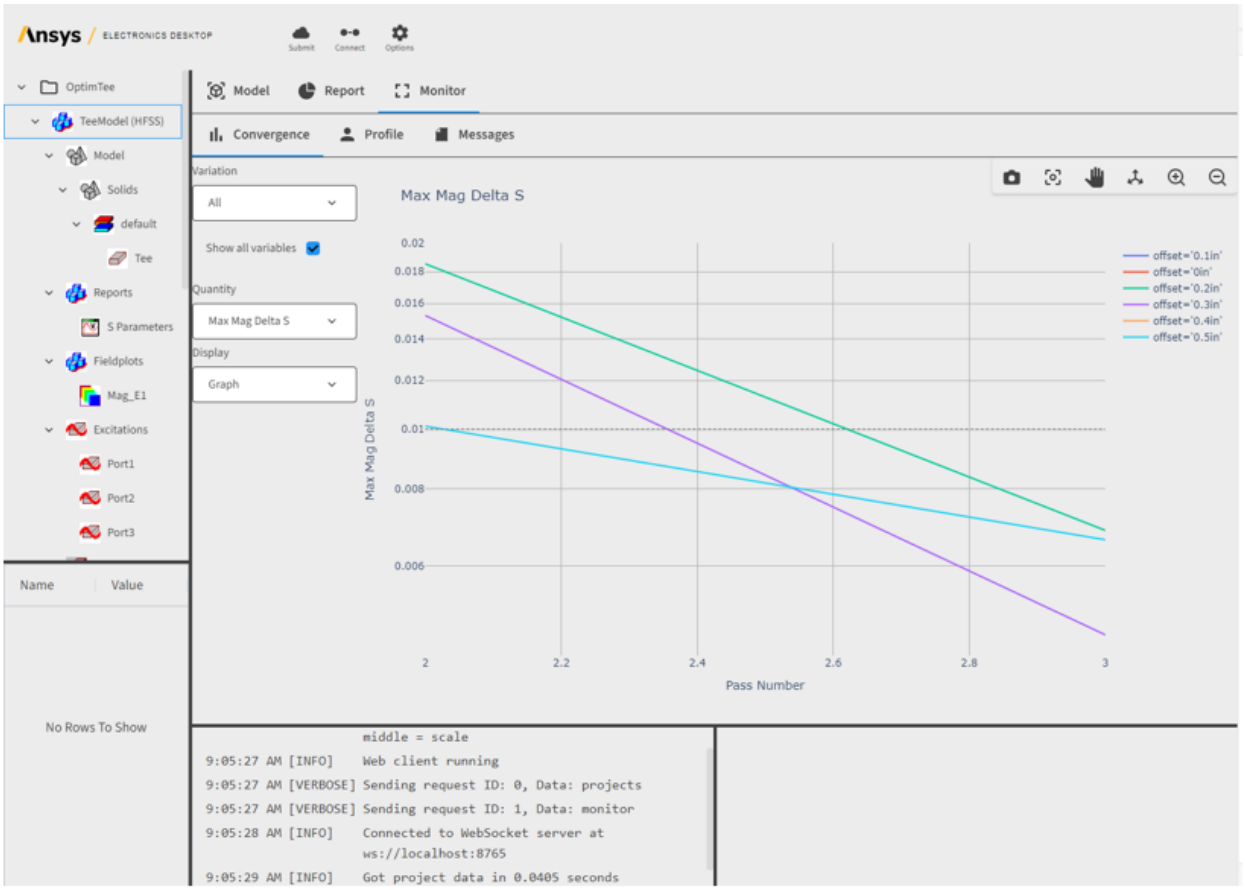
Click **Open Web Client** to open the Web Client in your default browser.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



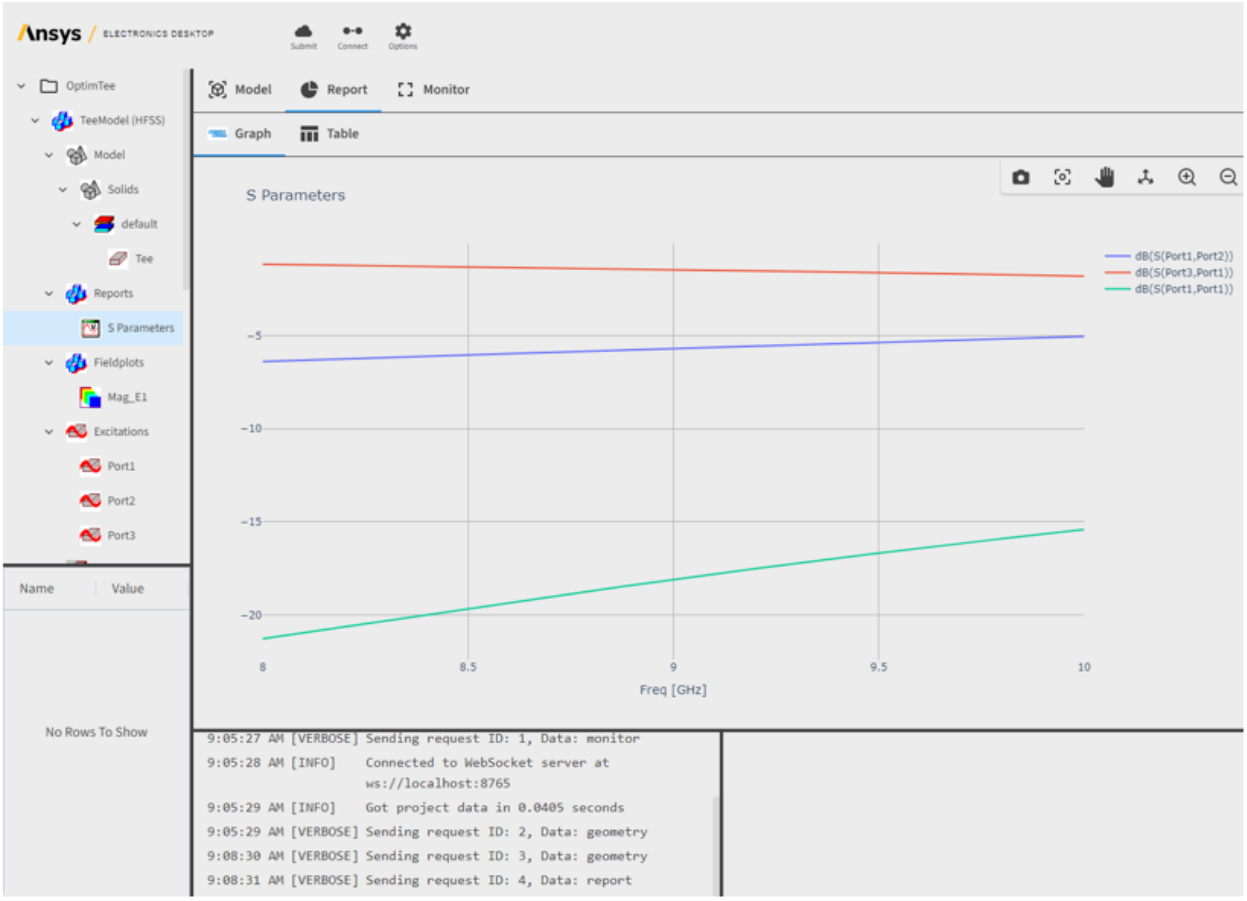
Switch to the **Monitor** tab to gain access to the subtabs for Convergence, Profile and Messages.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



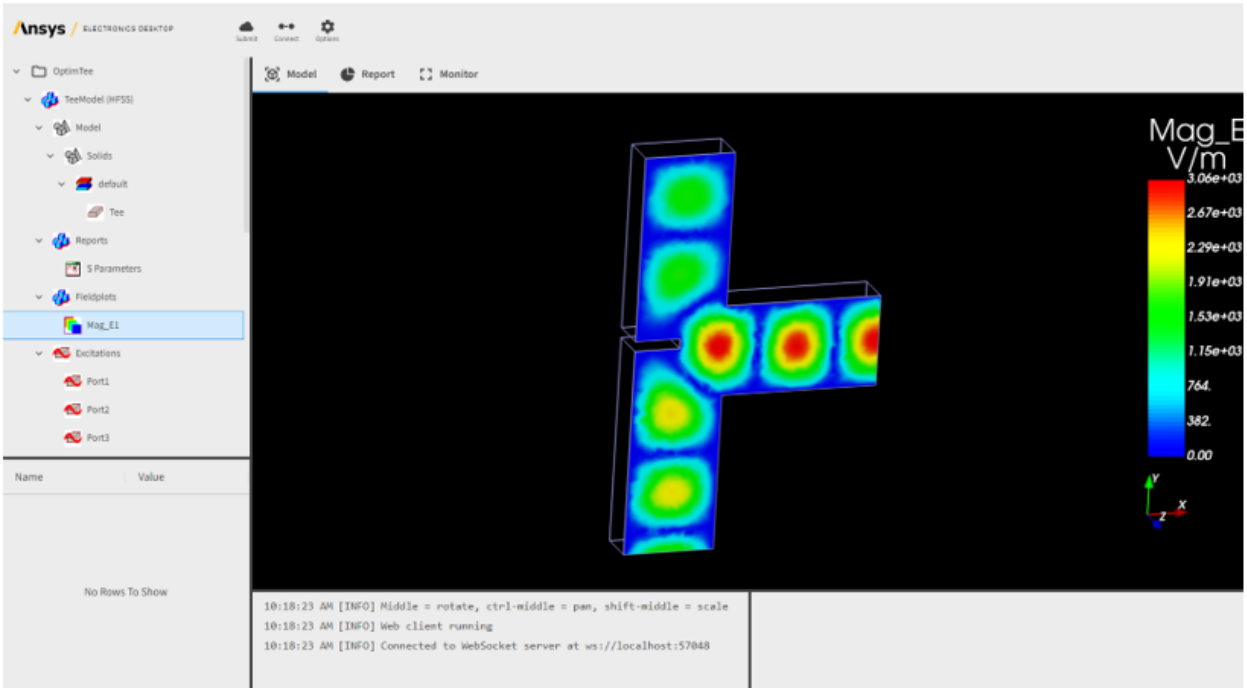
Select a Report in the **Project Tree** to view it.

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.



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?

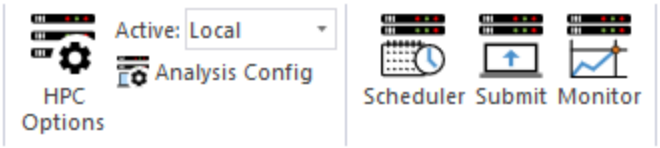
RSM is generally required if you want to run remote or distributed simulations. However, if you have a separate scheduling system that Ansys Electromagnetics supports, and you plan to run batchsolve simulations only, you may not need to install RSM. For details of installation and configuration of RSM, see the *Ansys Electromagnetics Installation Guides*.

Job Management UI for RSM

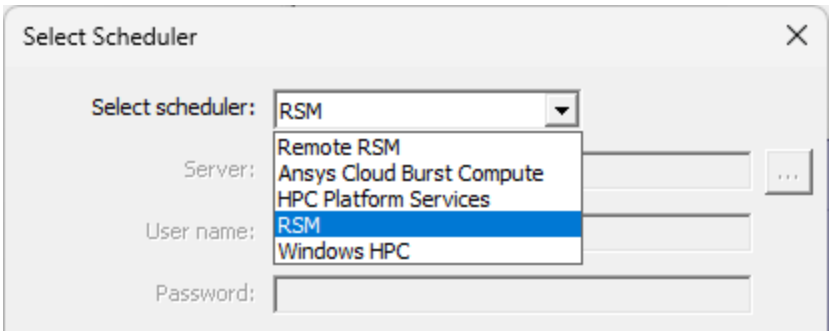
You can use the Job Management UI to submit batch jobs to RSM. The Job Management UI is accessed by running Ansys Electronics Desktop on the designated 'Postprocessing node' of the cluster. The Desktop provides UI commands for Scheduler selection, Job submission and Job monitoring/control. You access the Scheduler User Interface by clicking **Tools** > **Job Management** > **Select Scheduler...**

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 of the ribbon, and click the **Scheduler** icon.



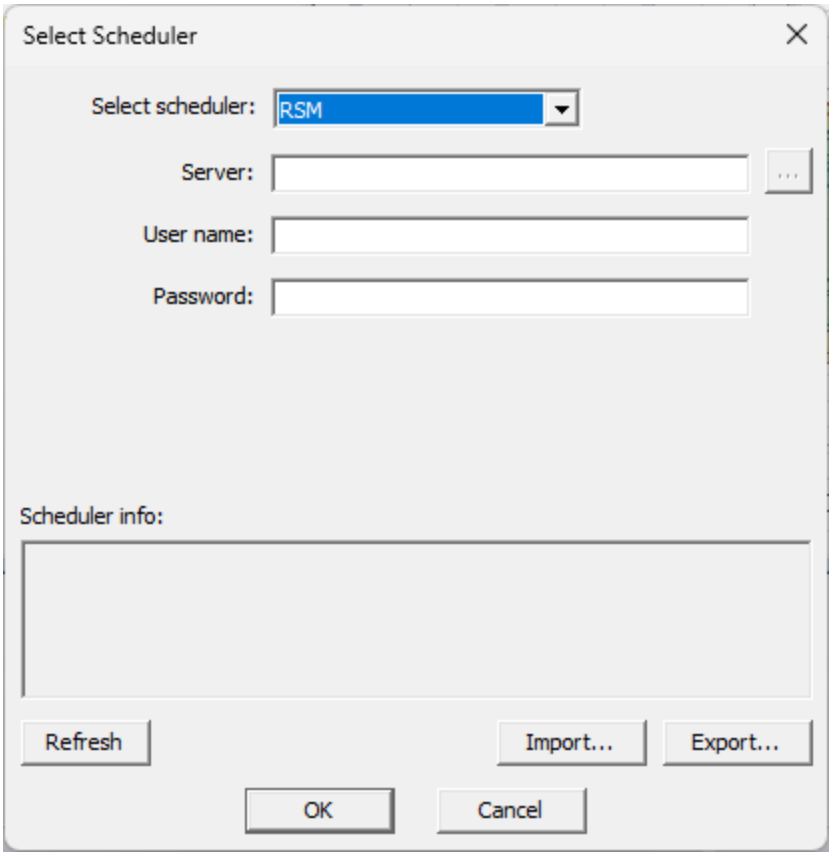
This displays the selection dialog box. The **Select scheduler** drop-down menu lists potential schedulers (list is defined by your environment).



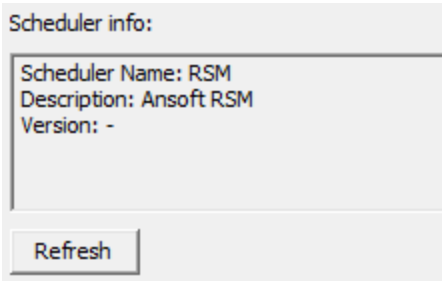
If you select a scheduler that is not supported in your environment, you receive a warning.

If you select Remote RSM and your environment has been configured, you can select a computer, user name, and password.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After selecting a scheduler, click **Refresh** to display information for that scheduler.



Once you have selected a scheduler supported in your environment, you can go through the following steps to submit a batch job.

1. Set up and prepare model on local workstation.
2. Copy the input project (or folder, if the project references external files) from a personal workstation to a shared-drive on cluster (say project is copied to /home/projects/spool/test.adsn).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the RSM environment, you are required to specify a machine-list. See: HPC and Analysis Options. For example, say the machine-list is: 3 cores from 'm1' and 3 cores from 'm2', for a total of 6 engines. You select the list on the **Compute Resources** tab of the **Submit Job to RSM** window, as described below.

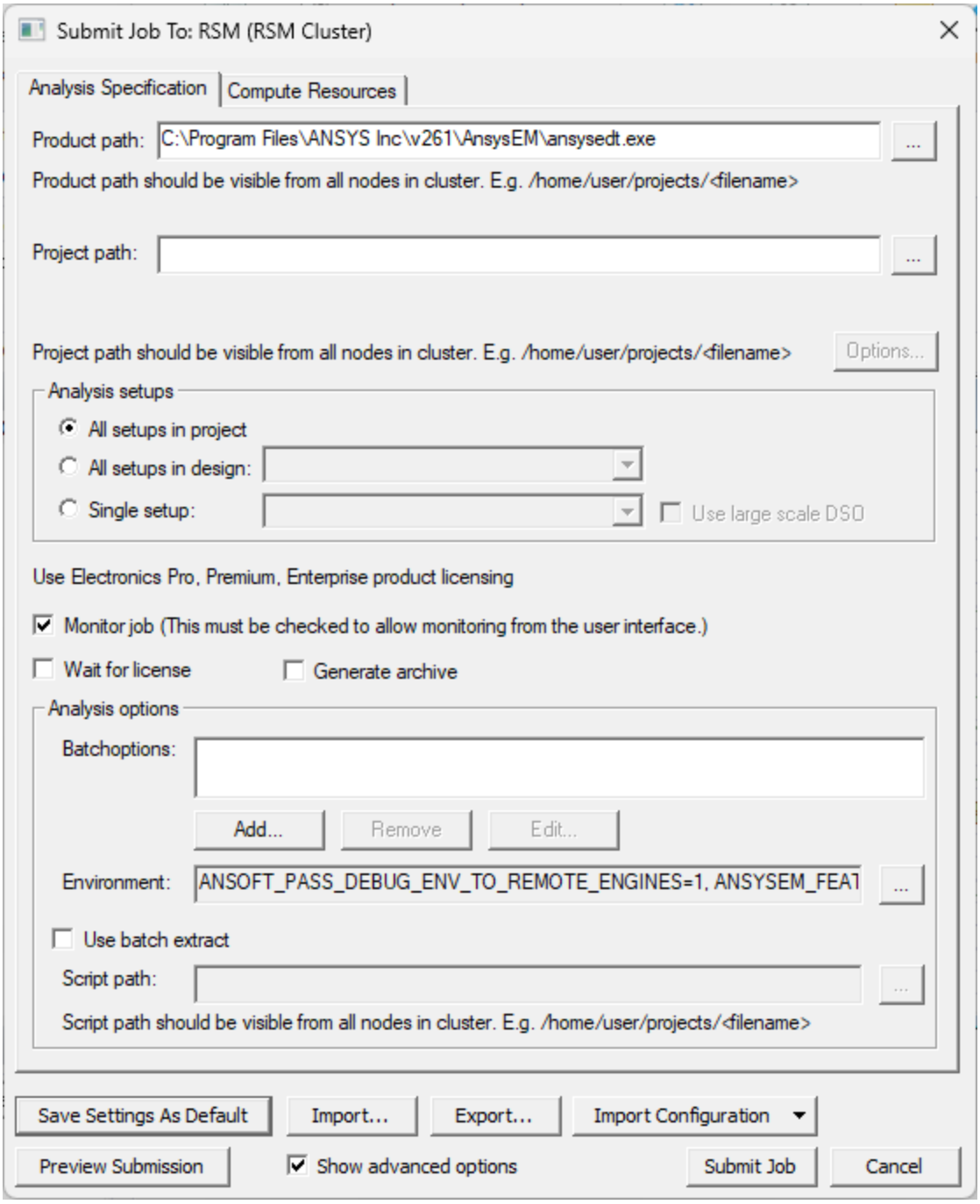
3. Open a remote-desktop session (or equivalent such as vnc session) on the node corresponding to the first machine of job's machine-list, 'm1' in this case. Launch Desktop graphically on 'm1'.
4. After setting the job submission node, select **Tools** > **Job Management** > **Submit Job...** or **Project** > **Submit Job...** or **[ProductName]** > **Submit Job...** to open the **Submit Job To:** window. You can also access **Submit Job** from the shortcut menus for the Project Name, Design name, Analysis Setup, or Optimetrics Setup.

   The **Submit Job To:** window contains two tabs:

   - **Analysis Specification** – specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
   - **Compute Resources** – specify the amount of compute resources and how to select specific resources from those available, and automatic settings, if supported by the design types.

   The standard Job Submission window displays.

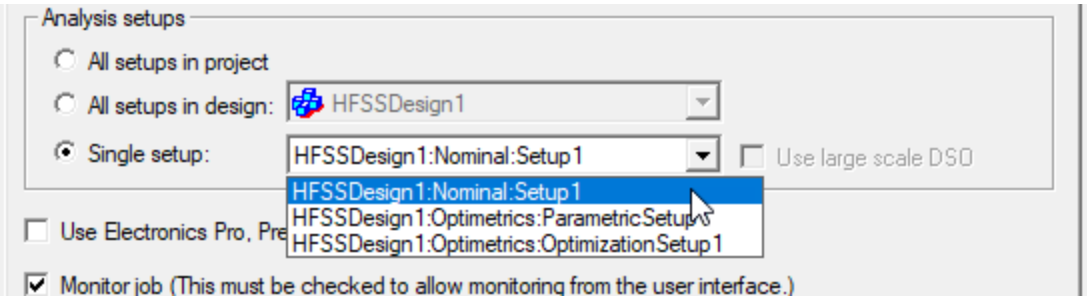
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

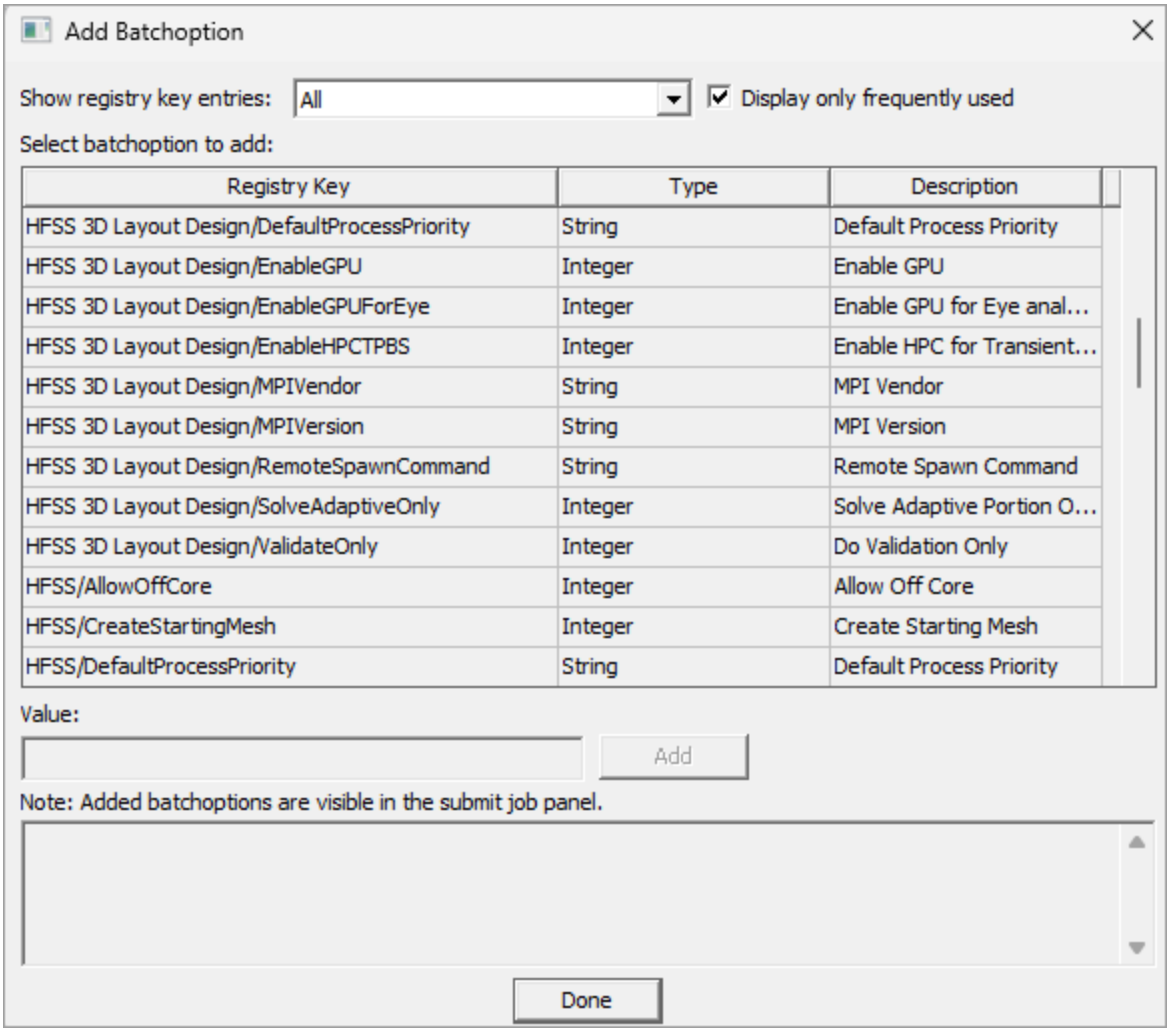


For Parametric setups, you have the option to select **Use Large Scale DSO**. For details on how and when you use this feature, see Job Management Interface for Large Scale DSO.

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

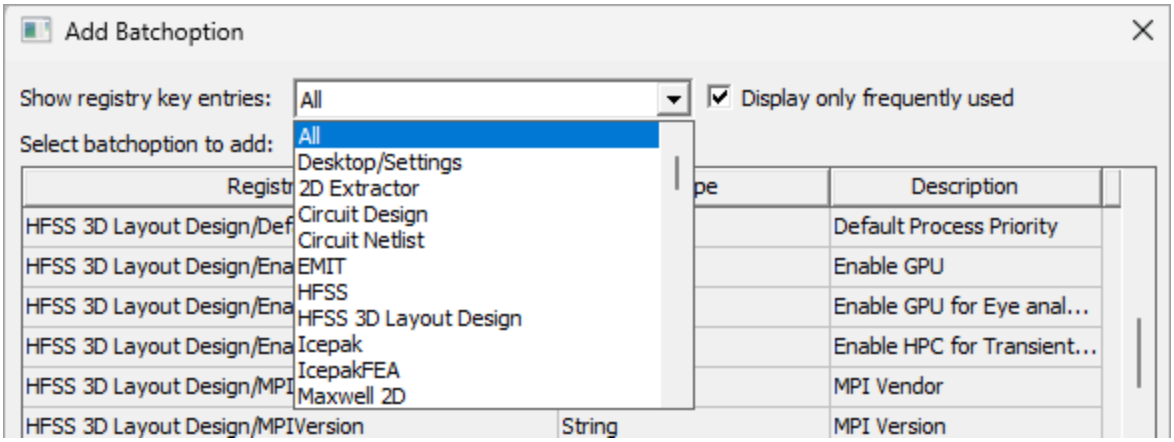
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menus. Click the **Add** button to view the **Add Batchoption** dialog box.

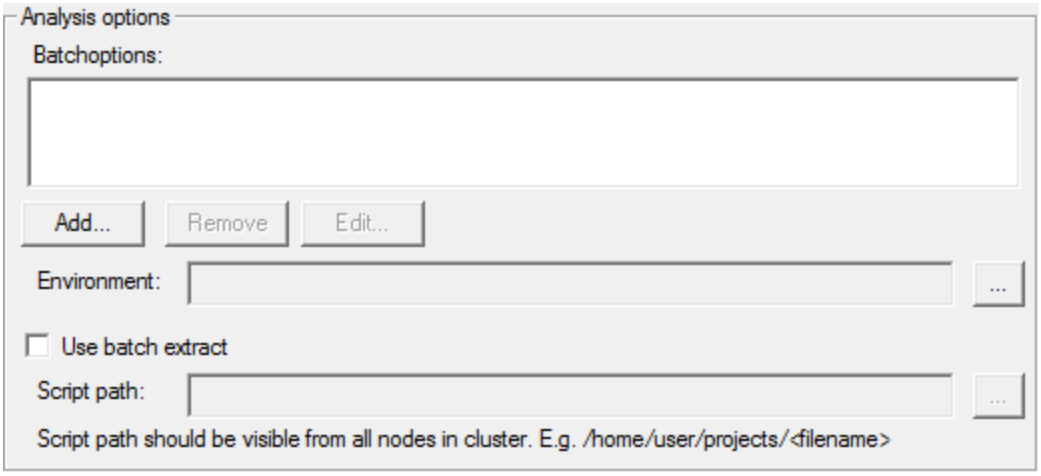


The **Show registry key entries** field lets you filter the entries displayed, by means of drop-down menu selection, and a check box to **Display only frequently used entries**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

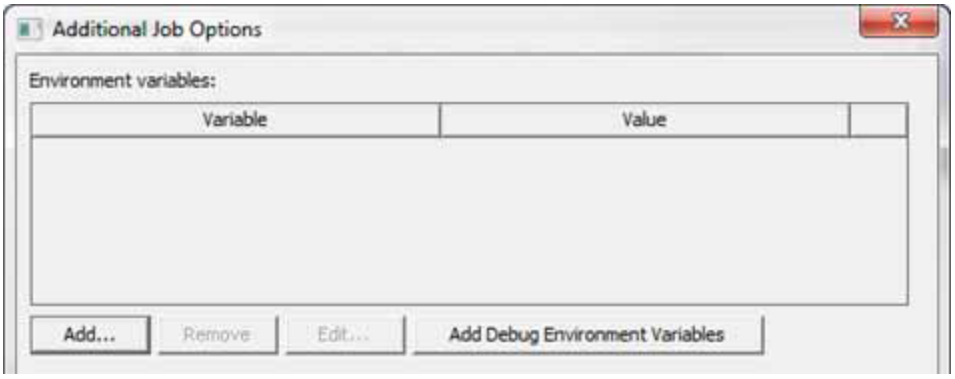


- When you have selected a batch option, you can type the value in the field, and click the **Add** button to add the option to the batch command.
- In the **Submit Job To:** window, you can enable **Show advanced options** to display additional fields for environment variables, and to display **Use batch extract** setting.

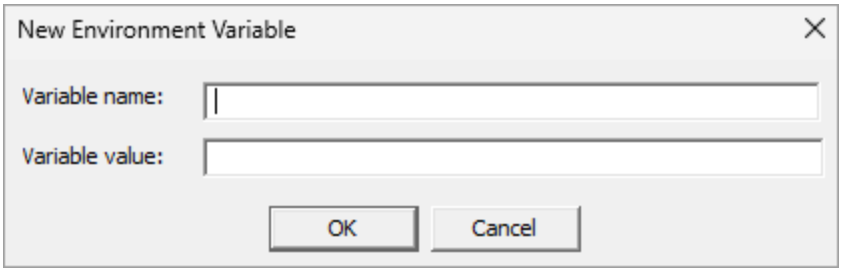


The Environment field is for environment variables, for instance, for debugging features or other variable controlled features. Click the ellipsis [**...**] button to open a dialog box for **Additional Job Options**.

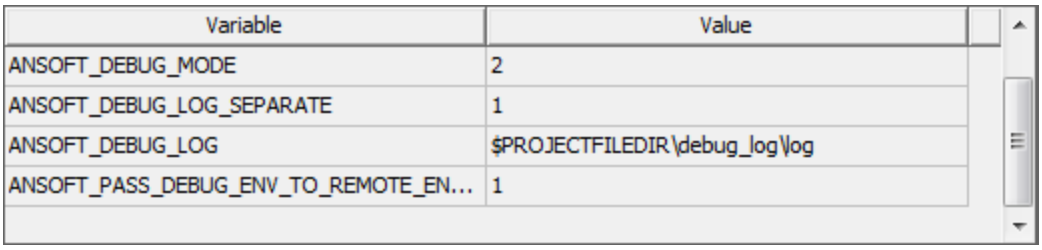
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Add...** button opens a **New Environment Variable** dialog box in which you can include a variable name and value.



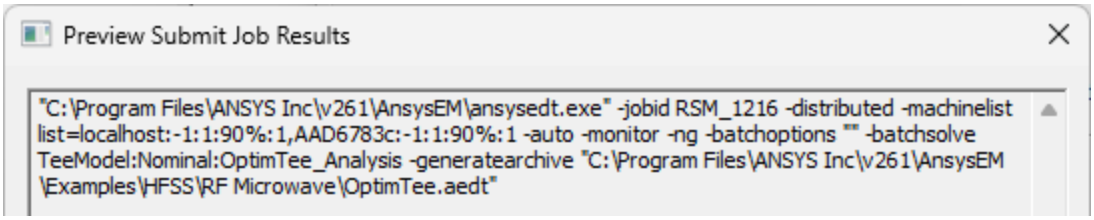
Clicking the **Add Debug Environment Variables** button automatically adds a set of debug variables. This can be useful in working with Ansys Application Engineering support.



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.

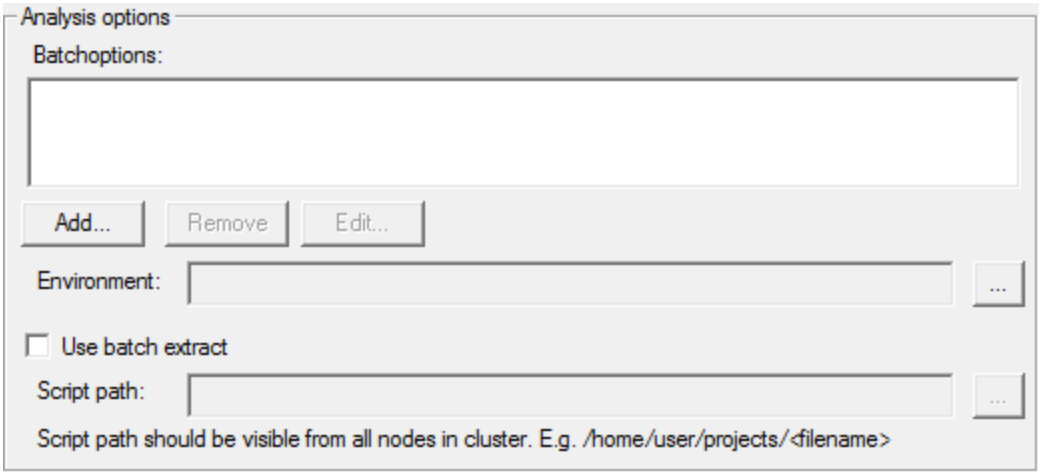
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The text can be copied to the clipboard, if desired.

Use Batch Extract for RSM

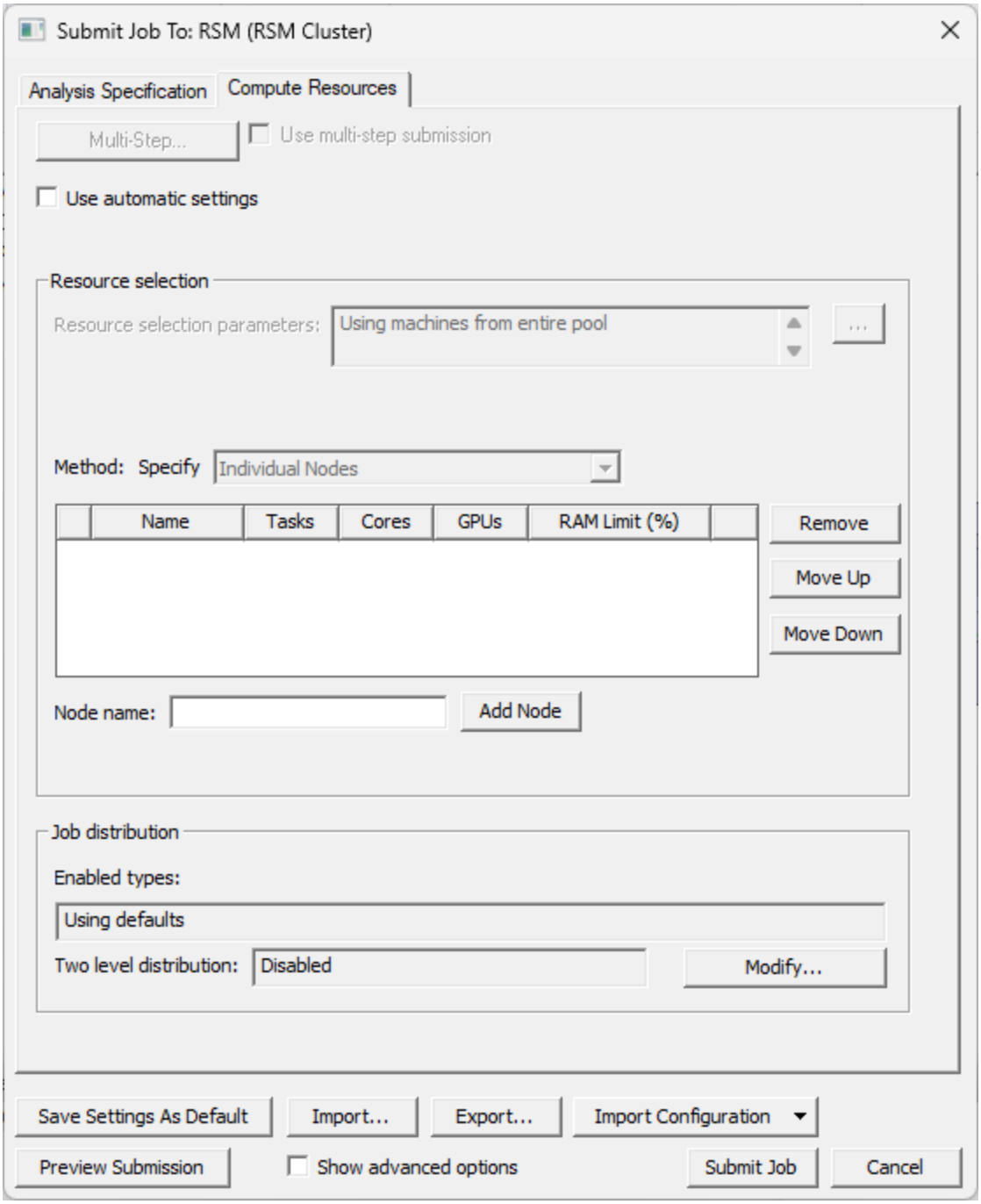
Selecting **Show advanced options for RSM** also shows the Use batch extract fields.



See the discussion on Running Ansys Electronics Desktop from a Command line for a discussion of the solve information available through batch extract.

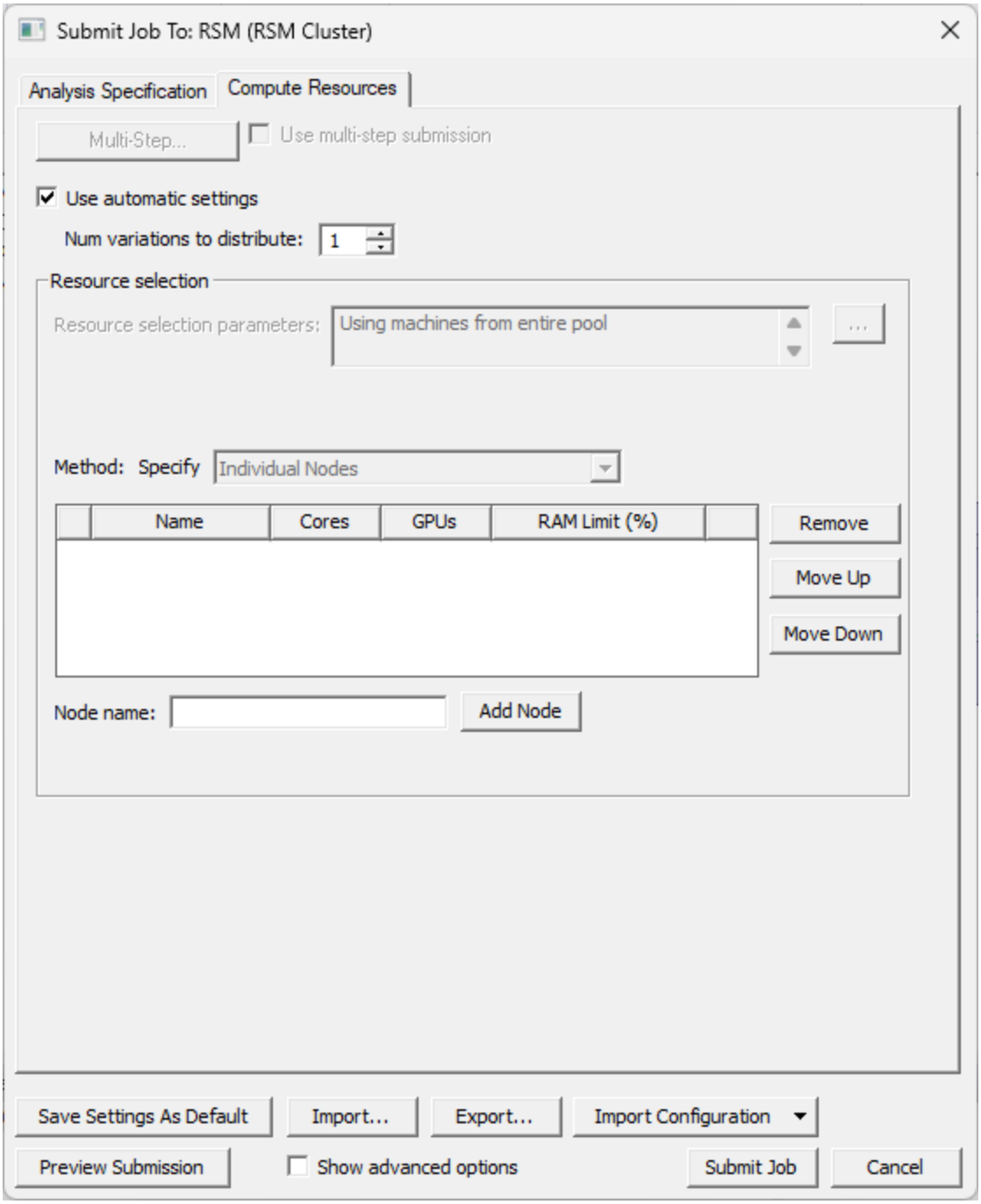
9. The **Compute Resources** tab of the **Submit Job to: RSM** window displays other parameters. Depending on the resources available for a scheduler environment, some of the fields may be disabled.

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.

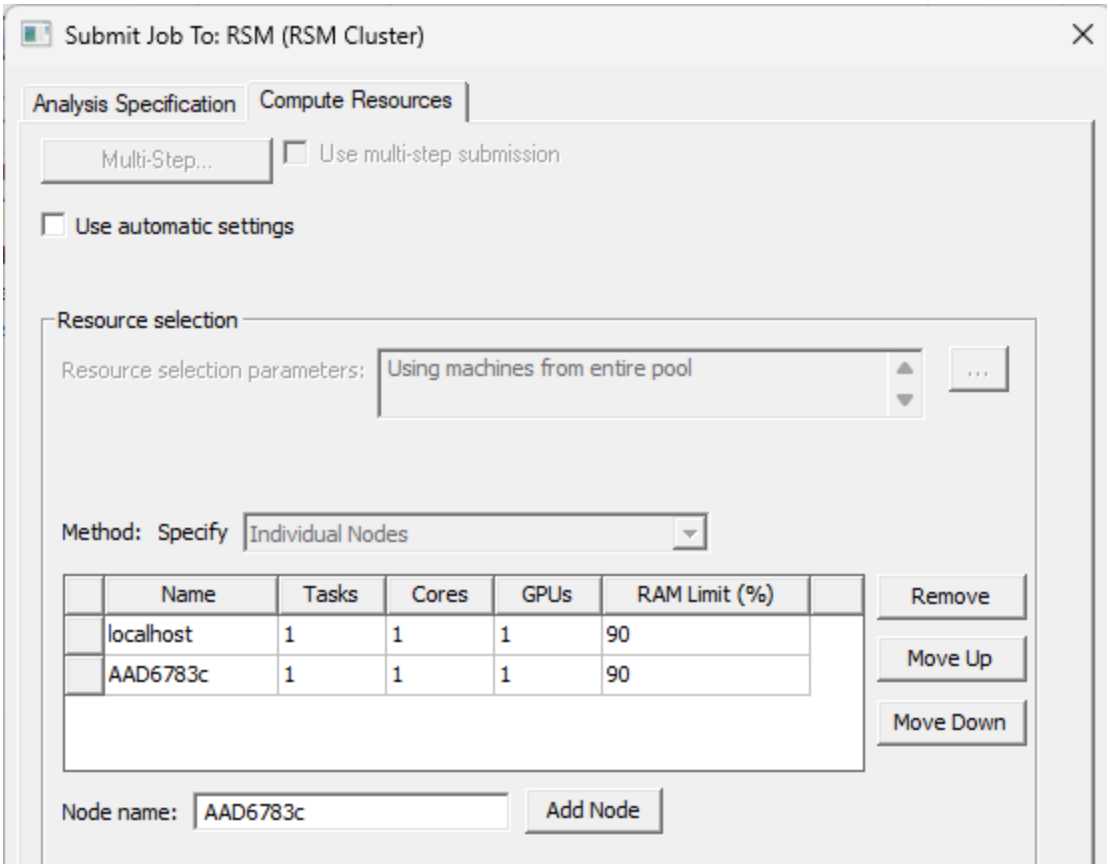
Note:

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.

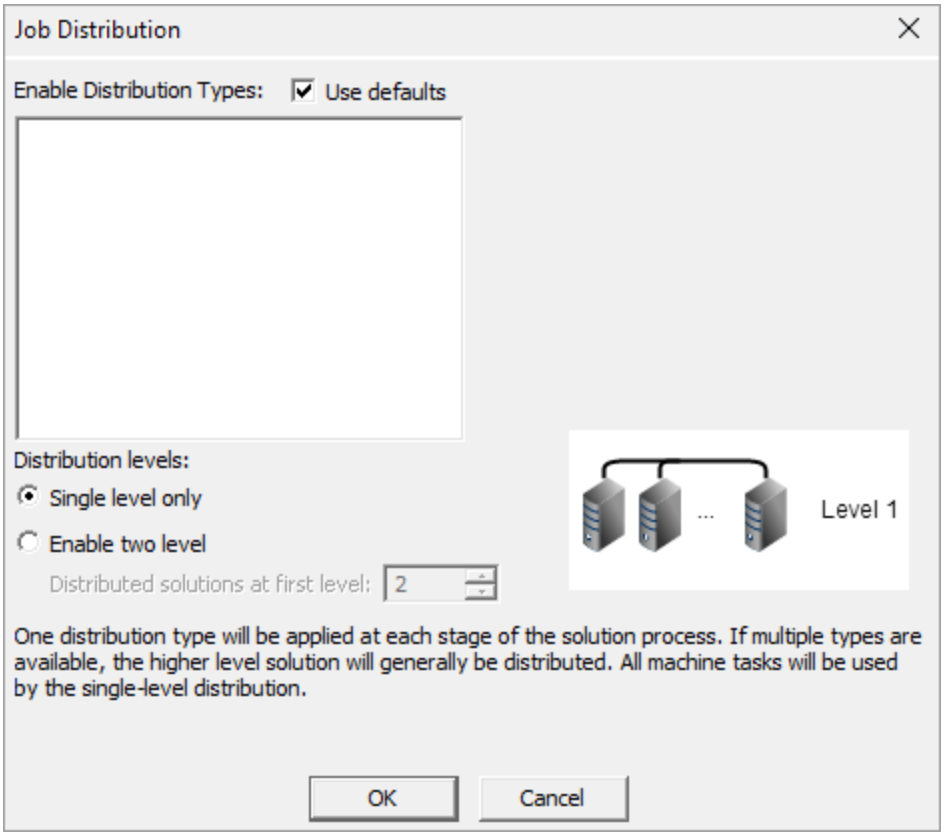


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job Distribution

If you disable User automatic selection, you can modify the Job distribution settings.

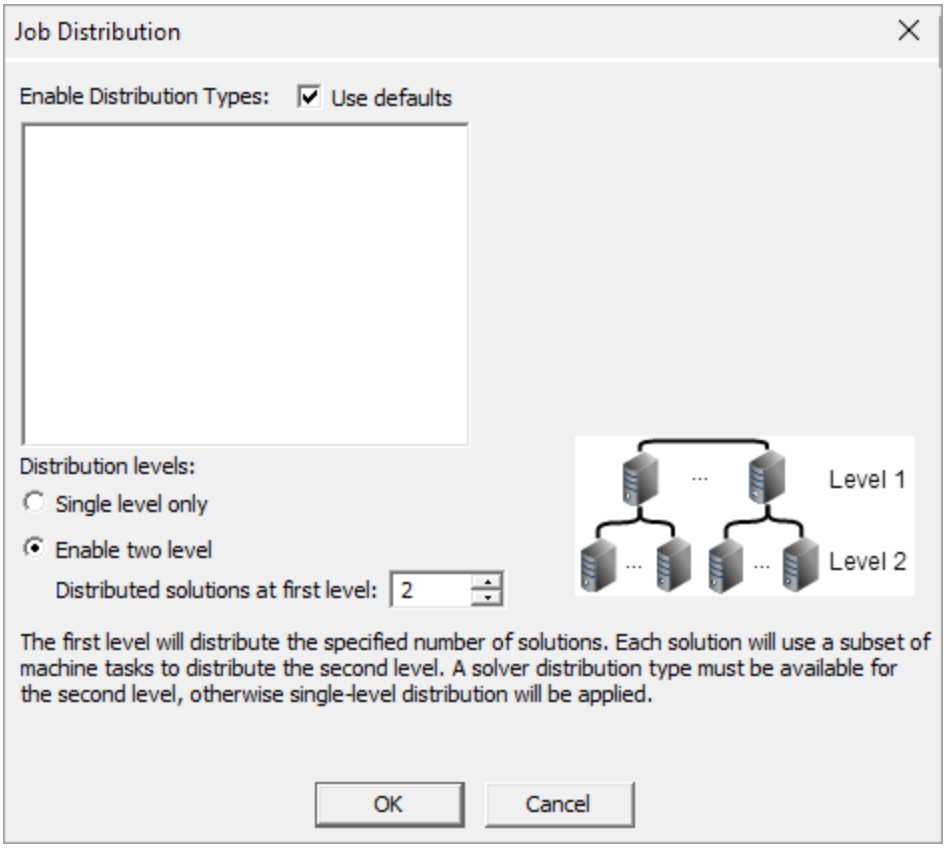
- Single level or two level distribution (*single level* is the default). Click **Modify** to display the *Job Distribution* dialog box and select the **Enable two level** option if appplicable and desired.



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.



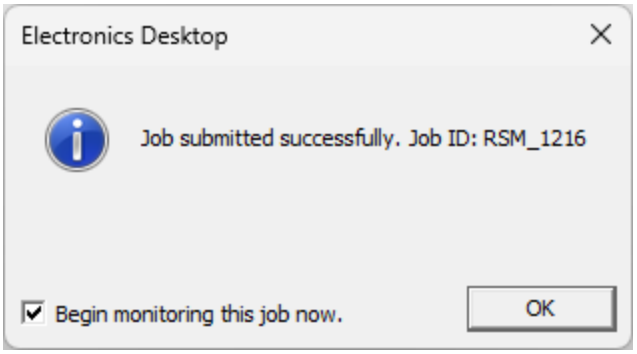
In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, 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'

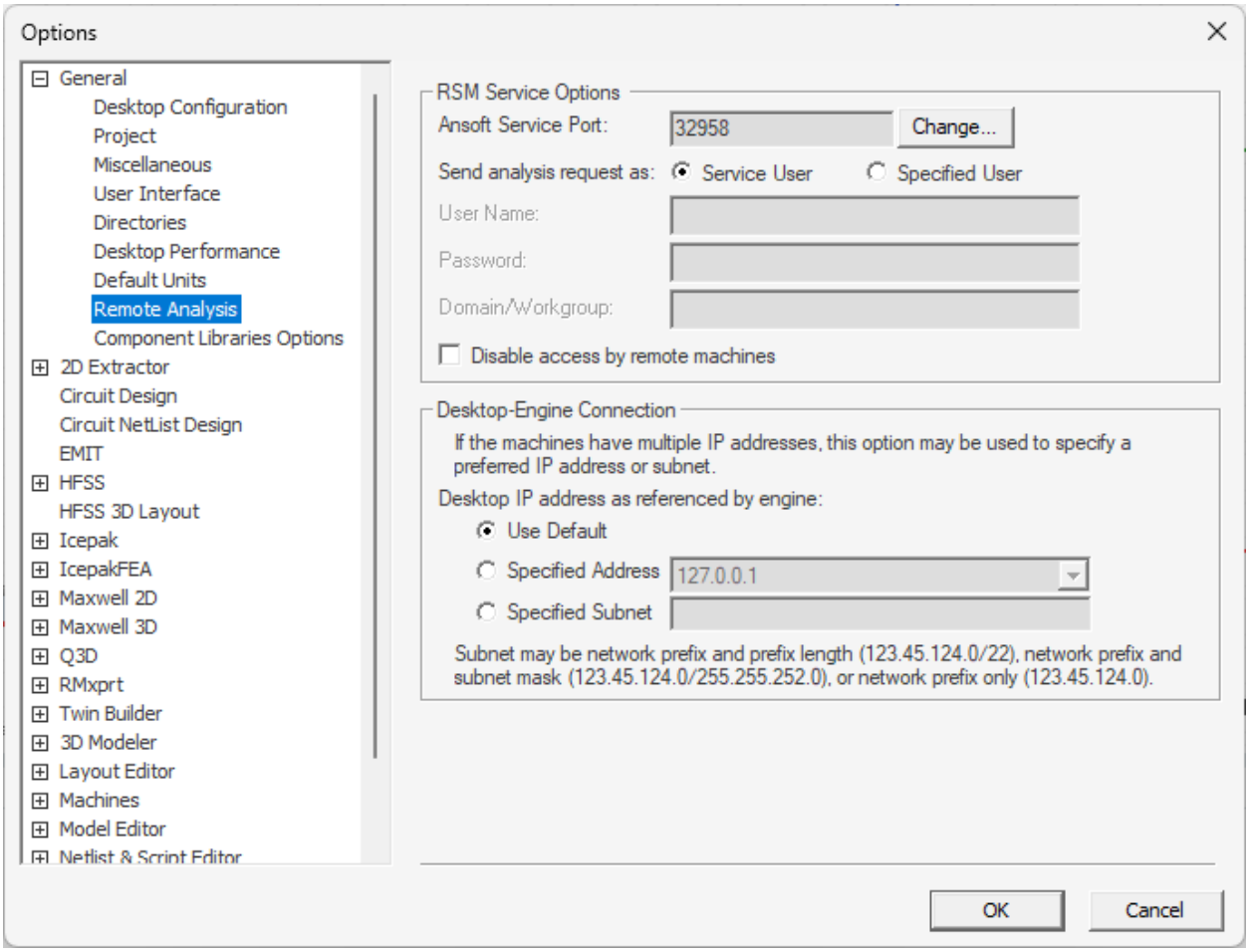
Ansoft Electromagnetics Desktop Configuration

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**.

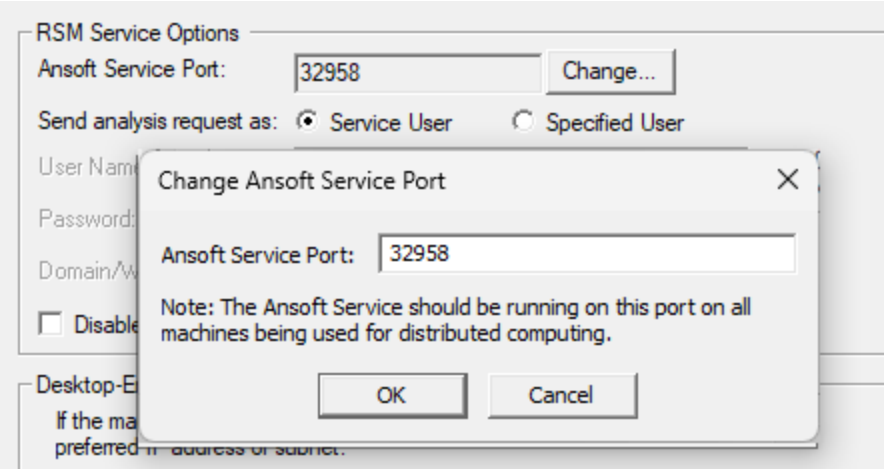
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

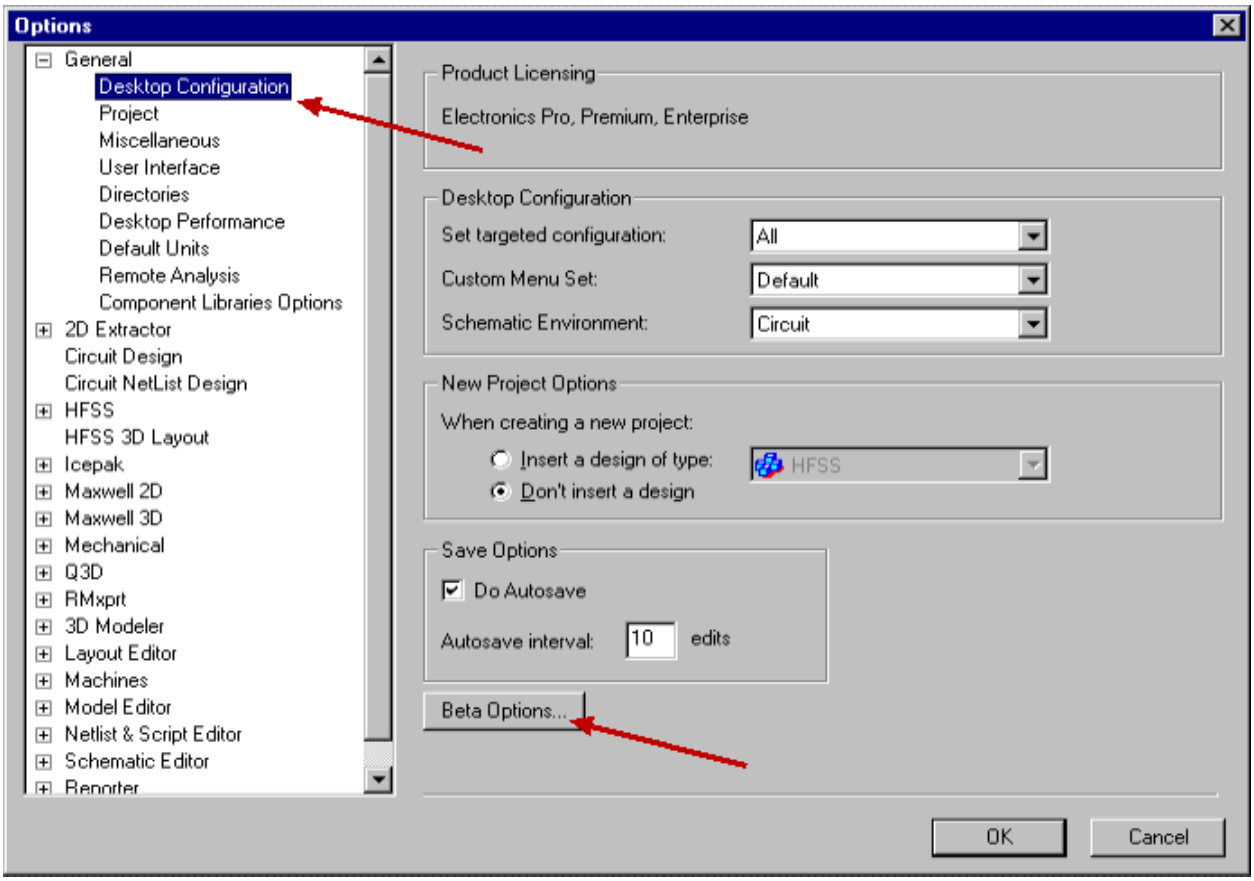
Support for Open MPI on Linux [Beta]

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.

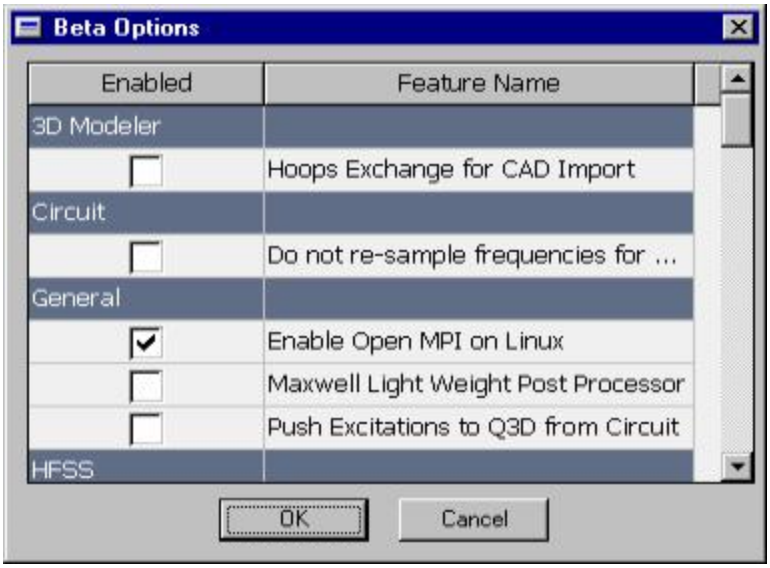
To enable the beta feature:

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.

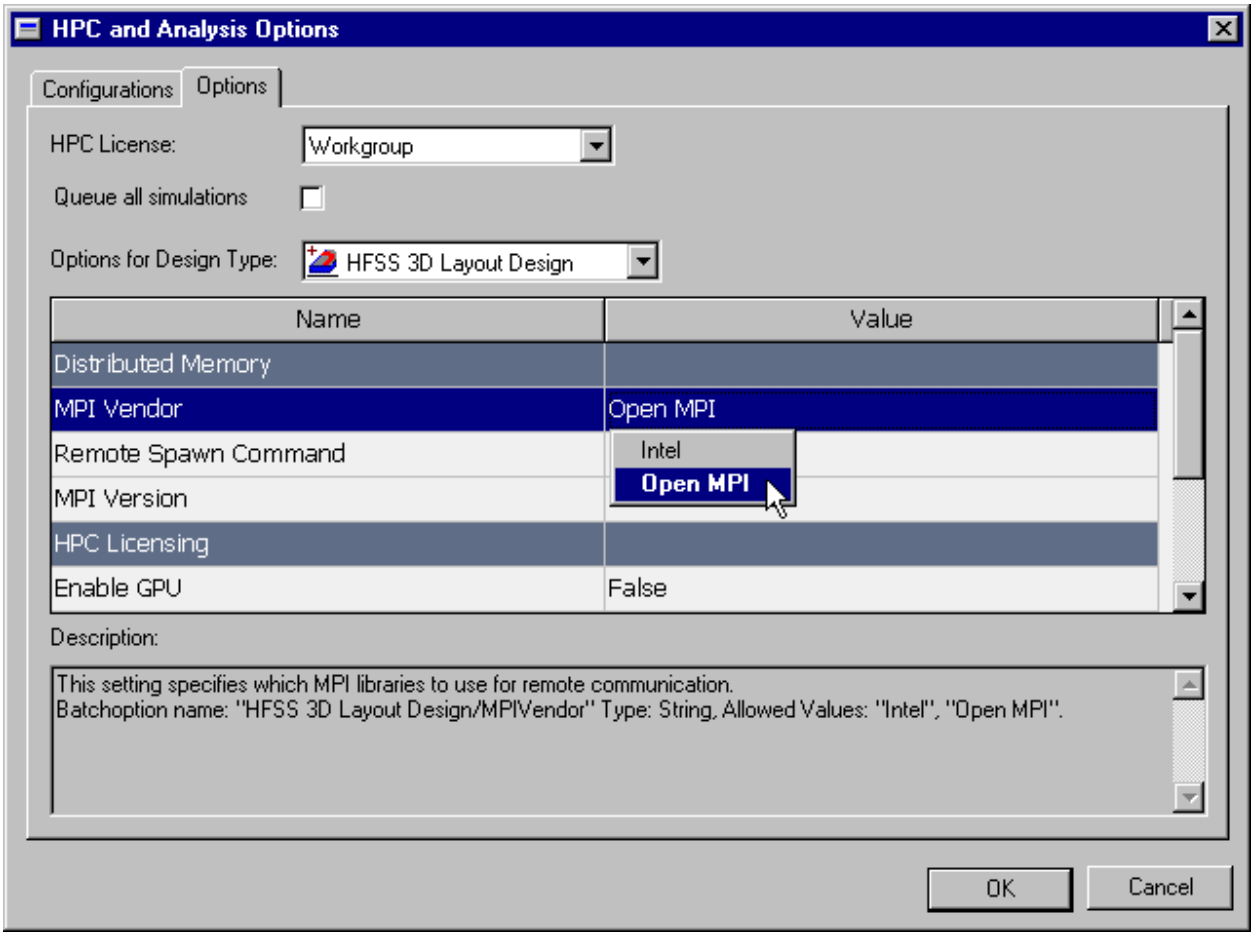
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

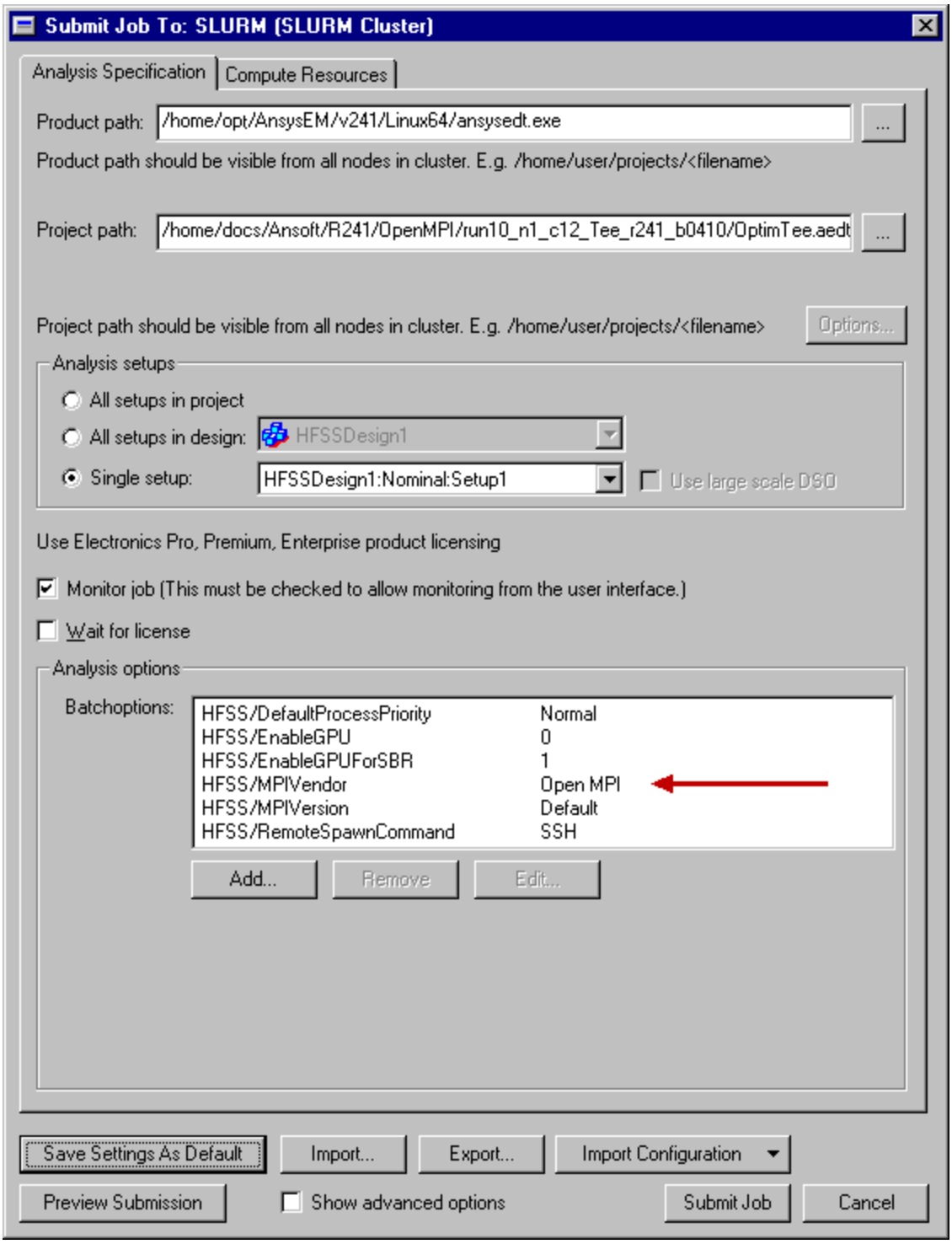
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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, 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,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

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 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 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

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**
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.

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.

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** 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**.

Aborting an Analysis

Opening the Abort Job menu

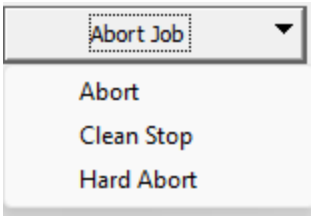
The **Abort Job** pop-up menu can be opened by any of the following options:

- 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 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 hfss.exe.

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 - Circuit and Layout Definition Libraries

This topic describes how to configure and manage **Electronics Desktop** definition libraries. You can use these topics to create and edit circuit and layout library elements.

The typical Electronics Desktop user requires these library tools only rarely. The Desktop automates the creation of many kinds of components. This topic covers only the operations needed to create and maintain components like the built-in Nexxim and EM design components.

- For information on creating component models, see Circuit and Layout Import and Export Operations.
- For information on creating components to be simulated via dynamic links and Solver on Demand, see Co-simulation.
- For information on creating and modifying substrates for use by the layout field solvers, see the *Nexxim Component Models* topics for each kind of distributed element (coplanar waveguide, grounded coplanar waveguide, microstrip, offset stripline, rectangular waveguide, slotline, stripline, and suspended stripline).

Related Topics

Working with Definition Libraries

Using the Library Editor

Using the Material Editor

Using the Script Editor

Using the Component Editor

Using the Padstack Editor

Using the Symbol Editor

Using the Footprint Editor

Vendor Library Components

Encrypted Libraries

Working with Definition Libraries

In Electronics Desktop, a component is a object placed in a design to be solved. Built-in Ansys components contain netlist and footprint information representing the native circuit and EM solver models. Other components can reference external models using frequency-dependent network data or other model formats.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The definition of a built-in component contains parameter data to pass to the solvers. The component definition also references separately stored data for the schematic symbol, fabrication material, layout footprint, and layer stackup definitions. These component dependencies are stored as objects in their own definition libraries.

System, User, and Personal Library Directories

The stock 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.

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 foundry support, user-defined models, and any custom or proprietary sets of components or simulation models. Customarily, **userlib** is a network repository for proprietary or corporate definitions available to all seats in an enterprise. **personalLib** contains project and circuit-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.

Changing the Locations of Libraries

You can relocate the base library directory to a new valid library location. To specify a new library directory:

1. Click **Tools** > **Options** > **General Options**.
2. In the **Directories** group in the **Project Options** tab, type the new folder location in the **Library Directory** group box, or click **Browse ...** to specify it.
3. Click **OK**.

To return the library directory specification to its default value, click **Reset Library Directory**.

The personal library (**PersonalLib**) folder is located at **<Project Directory>\PersonalLib**, where **<Project Directory>** is the location you specified for your Electronics Desktop projects. To specify a new location for the **PersonalLib** folder, you must specify a new project directory location. To do this:

1. From the **Tools** menu, select **Options** > **General Options...**.
2. In the **Directories** group in the **Project Options** tab, type the new folder location in the **Project Directory** group box, or click **Browse ...** to specify it.
3. Click **OK**.

Library and Project Definitions

In Electronics Desktop, library component definitions are intended to be accessed once per component — when you place the first instance of a component onto a schematic. After it is

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

placed in a schematic, the definition for the component transfers on the library to the project file. Editing and updating a component definition is then controlled on the *project definition*, which is listed in the **Components** folder in the **Project Manager** window. In other words, you do not edit component definitions in the library, but in the project. If you want your edits to be reflected in the library definition (for use in another design) you must export the edited components to a component library.

Once a definition is used, it is transferred to the current project, and remains in the project unless it is explicitly removed. To see the definitions included in a project, expand its **Definition** folders and subfolders in the **Project Manager** window:

Modifying component and dependency definitions in libraries, or installing libraries with modified component and dependency definitions, does not automatically update those definitions in projects that contain them. To update project definitions from library definitions, see Updating Project Definitions from Library Definitions.

Updating Project Definitions from Library Definitions

Once a library component is placed into a design, the parent component definition resides in the project, and the link to the library is broken. To update a project component definition with a definition from a library file, select the project you want to update in the **Project Manager** window, then click **Project Tools** > **Update Definitions** on the **Tools** menu.

The **Update Definitions** window opens, listing any library definitions that have been edited since they are added to the project, and their original library locations. Use the window to select the item to update, and click **Update** to finish.

If you wish to remove project definitions that are unused, choose the **Project Tools** > **Remove Unused Definitions** command on the **Tools** menu which opens the Unused Definitions window.

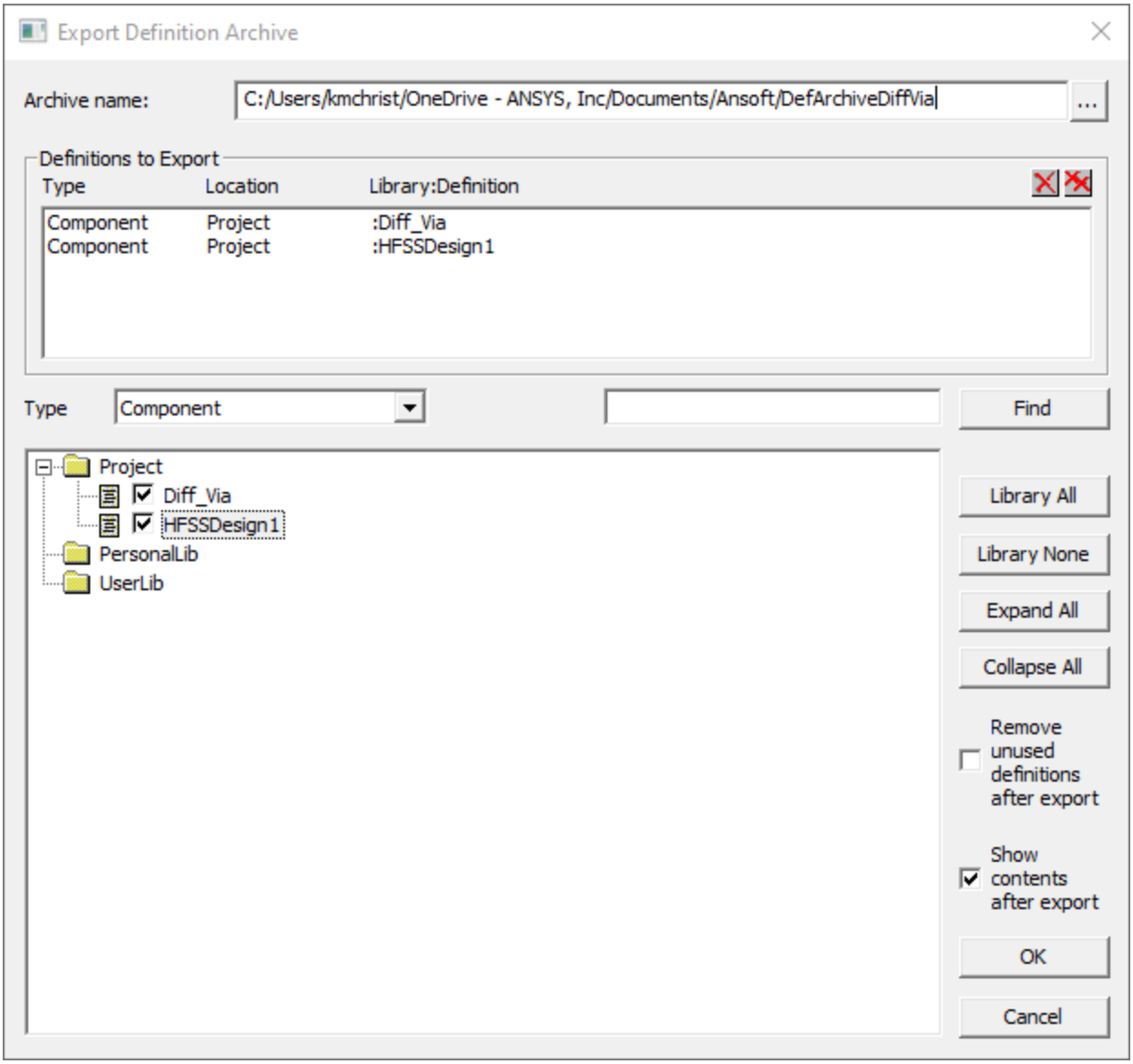
Exporting and Importing Definition Archives

The Import and Export Definition Archive commands allow you to select individual component, model, or other library definitions – or even entire libraries – to package and send to another user. Referenced definitions, data files, and DLLs are included automatically. All selected definitions (either those in an entire library or those selected separately) is written to one set of libraries (e.g., a component library and associated symbol and model libraries).

Exporting Definition Archives

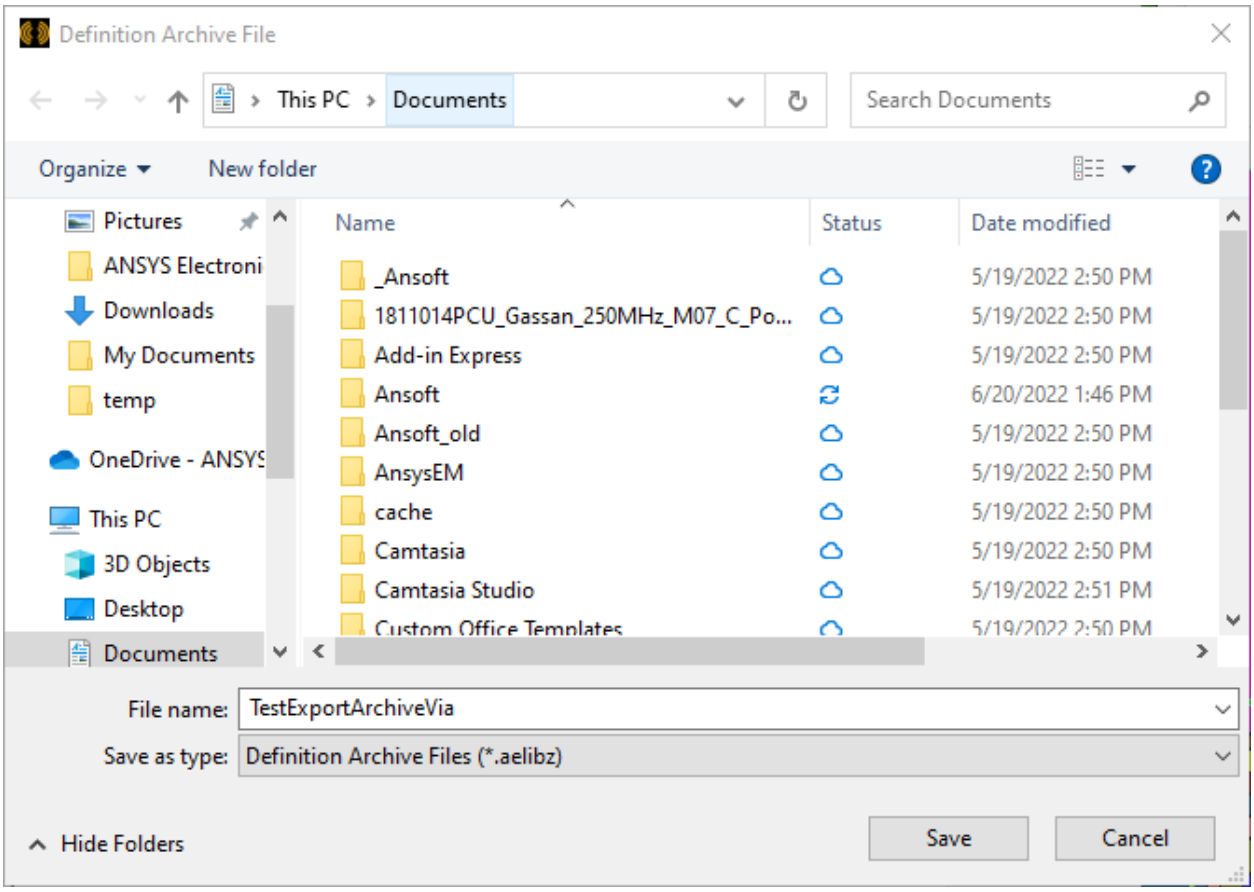
Select **Tools** > **Library Tools** > **Export Definition Archive** to open the **Export Definition Archive** window.

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 a definition archive, first choose an archive name, either by typing the full filename in to the **Archive name** edit field, or by using the **[...]** button. This button opens 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.



- The archive name must have a Definition Archive Files **.aelibz** extension, which is supplied by the window.

> **Note:** An **.aelibz** archive is not readable with utilities such as WinZip®.

- The definitions selected for export are listed in the **Definitions to Export** list box. Click the and buttons to delete selected entries, or all entries, respectively.
- The **Type** drop-down menu on the left, under the list of **Definitions to Export** controls which libraries are shown in the tree under it. Choices include: **Component**, **Symbol**, **Model**, **Package**, **Material**, and **Script**. In the tree, each leaf element (definition) has an associated check box. Checking the box adds the definition to the export list above. Clearing the box removes the definition on the export list. With a library or library element node highlighted, a user can select all definitions in that library for export by clicking **Library All**. Similarly, the user can unselect all definitions in the selected library by clicking **Library None**. **Expand All** and **Collapse All** expand or collapse the entire tree.
- Typing a string into the field next to the **Find** button allows the user to scroll to the first definition whose name matches the string. Searches are not case-sensitive. You can also use the “wildcard” asterisk character in search strings. For example: entering P* should

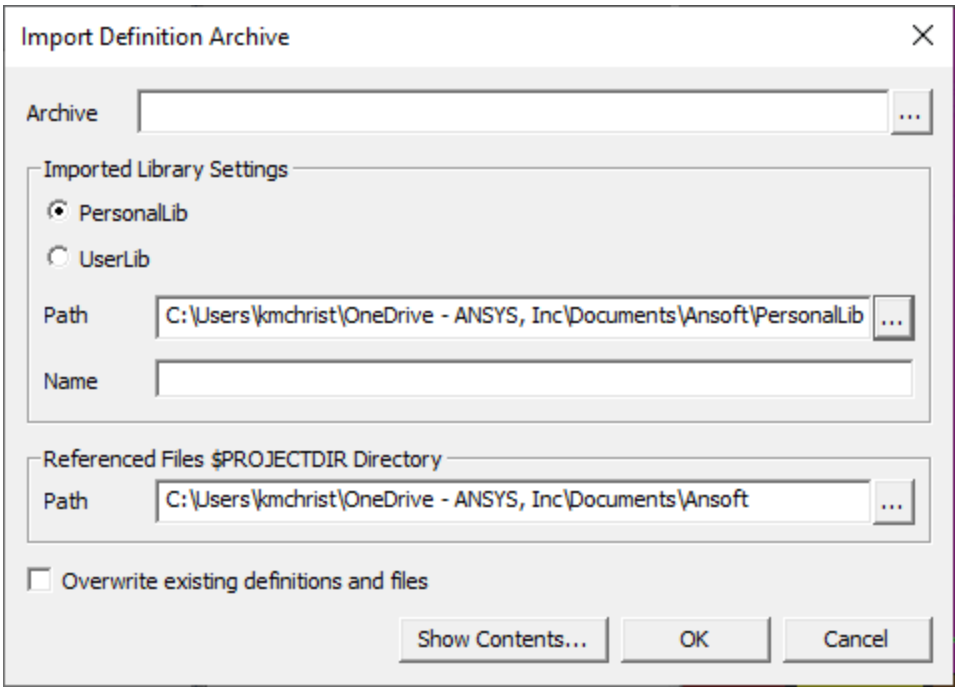
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

find, in succession, every component that starts with P (or p). Use multiple asterisks to further control matches (e.g., type *p* to find any definition with a 'p' in it, including as a first or last character; or *p*p* to find "pipe" and "pump", et cetera).

- The **Remove unused definitions after export** check box allows the user to clean up the current project after export is complete.
- When the **Show contents after export** box is checked, a **Contents** window shows a list of what was saved to the archive. This list may include multiple libraries (due to definition references) or external file references. Save the list to a file by clicking **Save , or** click **Done** when you are finished viewing.
- Click **OK** to complete the process. The selected definitions, libraries, and referenced files are zipped, and given a Definition Archive FIles extension, **.aelibz**. Users can then e-mail or otherwise transfer the archive file to others. The archive file is not encrypted, though models and packages contained in it may be.

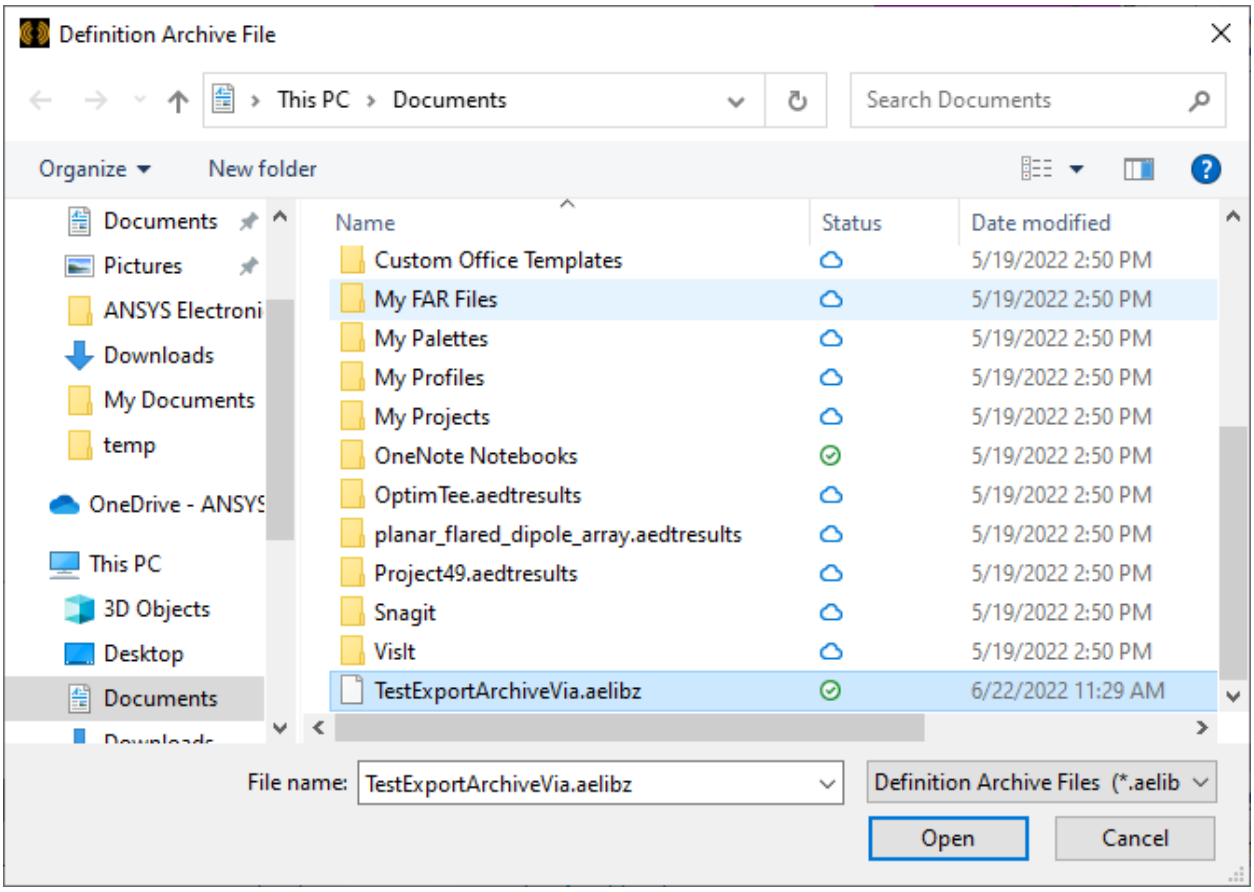
Importing Definition Archives

To import a definition archive, choose **Tools** > **Library Tools** > **Import Definition Archive** to open the **Import Definition Archive** window.



- First select a definition archive either by typing the path and definition archive file name in the **Archive** field, or click **[...]** to browse for the Definition Archive File created using the **Tools** > **Export Definition Archive** command.

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 **Imported Library Settings** group box, click **PersonalLib** or **UserLib** to choose the basic location for the imported archive. If appropriate, the user can further choose a subdirectory **Path** under **PersonalLib** or **UserLib** as a location for the libraries. The user can also enter a **Name** for the imported libraries. The default name is the base name of the archive file.
- On import, data files is moved to appropriate **bin** and **data** directories. External **Referenced Files** that are not appropriate to **bin** and **data** library directories is referenced using **$PROJECTDIR**. The user can click the **[...]** button to choose the directory for these files. The default is the Project Directory specified in **Tools** > **Options** > **General Options**.
- The **Overwrite existing definitions and files** check box (unchecked by default) allows the user to replace current definitions and files with archive elements, permissions allowing.
- Click **Show contents** to view a window showing a list of what is in the archive. This list may include multiple libraries (due to definition references) or external file references. Save the list to a file by clicking **Save**, or click **Done** when you are finished viewing.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Managing Library Files

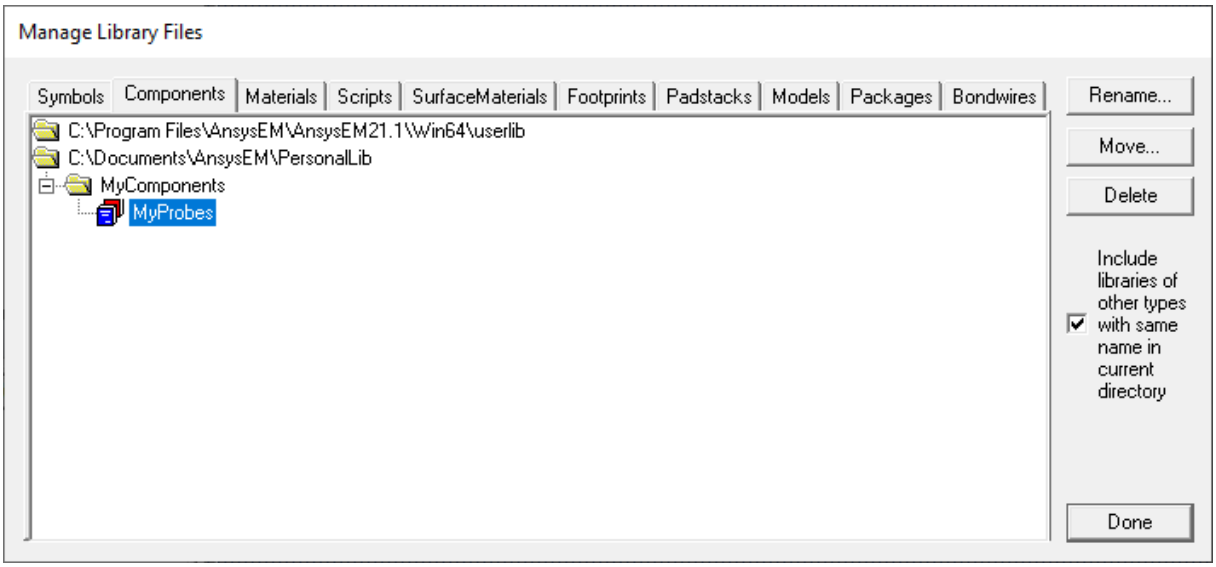
You can use **Manage Files** to change the names of libraries in the UserLib and PersonalLib directories; move libraries to another location in the directories specified for UserLib or PersonalLib, or delete libraries. You can also change the names of sub-directories of the UserLib and PersonalLib directories, move directories to another location in the directories specified for UserLib or PersonalLib, or delete directories.

> **Note:**
>
> - You cannot move libraries out of the UserLib or PersonalLib hierarchies.
> - To move a library or directory to a subfolder, that subfolder must already exist.

Components and other definitions in libraries directly affected by an action which See definitions changing library name or location has those references adjusted. Definitions in libraries indirectly affected– that is, those not having their names or locations modified by an action–with references to definitions in directly affected libraries are updated.

To manage library files:

1. Click **Tools** > **Library Tools** > **Manage Files**. The **Manage Library Files** window opens on the Components tab. The **Manage Library Files** window opens on the Components tab. There is a tab for each library type: Symbols, Components, Materials, Scripts, Surface Materials, Footprints, Padstacks, Models, Packages, and Bondwires.



2. Click the tab you want.
3. Select an individual library or a directory on which you want to perform the action.

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 **Rename**, **Move**, and **Delete**, as necessary.
5. Select the **Include libraries of other types with same name in current directory** check box to cause the action to be performed on other types with the same name in the same directory. For example, if you select “MyProbes” on the Components tab, the action is extended to a symbol library called “MyProbes.aslb” in the same directory.
6. Click **Done** when finished managing libraries.

Library Search Precedence

When you place a new component in a design, Electronics Desktop searches in the current project for a definition with the corresponding name. If an appropriate definition exists in the project, that definition is used to satisfy the placement *regardless of whether additional instances, even more recent ones, may exist in external libraries.*

If Electronics Desktop cannot locate the required definitions in the project, it searches in personal (**PersonalLib**), user (**userlib**), and system (**syslib**) library files, *in that order*. Once a definition of the correct name has been located, the search ends, and that definition is used to satisfy the placement request.

Component Creation Sequence

Every component definition uses an associated bitmap (for the **Project Manager** window icon) and a schematic symbol. Many components also require footprints, which in turn depend on material definitions and padstacks to characterize their physical presence in and electrical connections to a layout.

To save time in editing or creating new components, you should identify and modify or create its dependent elements in the following sequence as you define your component. All of the dependencies listed may not be necessary for every component you create or modify.

Materials

Material information defines the physical properties of a substance — such as substrate, metallization, or solder mask layer — for its inclusion in a *stackup*, the physical foundation of a manufacturable layout or model. Material definitions are stored in library files with the extension **.amat**. See Using the Material Editor for how to create and modify materials.

Layers

The substrate technology defines the name and parameters of the layout stackup to use for all distributed elements in the design. You create or define a substrate type by clicking **Circuit>Add Reference Data>Add Substrate Definition**. You can view and modify the current layout stackup by clicking **Schematic>LayourStackup**.

Padstacks

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A padstack defines the physical structure and electrical connectivity of a *pad*, the region where a component pin connects to a copper trace. A padstack must be associated with each pin of a component. Padstack definitions are stored in library files with an extension of **.pslb**. See Using the Padstack Editor for how to create and modify padstacks.

Footprints

A footprint defines the planar space consumption, physical orientation, metal usage, and electrical connectivity of a component in a layout. Footprint definitions are stored in library files with the extension **.aflb**. See Using the Footprint Editor for how to create and modify footprints.

Symbols

A symbol is graphical representation of a component in a schematic. Symbol definitions are stored in library files with the extension **.aslb**. See Using the Symbol Editor for how to create and modify symbols.

Bitmaps

A bitmap is a small picture that is displayed when the ACLB containing the component is opened in the schematic editor. Bitmaps are stored in syslib\Bitmaps*.bmp. To create or modify a bitmap, use a bitmap editor such as MS Paint(r).

Components

The Electronics Desktop **Component Editor** supports the viewing, modification, and saving of information for component elements and graphical primitives. After creating or modifying a component, the information can be saved, used to render a component, or exported. Component definitions are stored in library files with an extension of **.asty**. See Using the Component Editor for how to create and modify components.

Scripts

The Electronics Desktop **Script Editor** supports the viewing, modification, and saving of script file information for footprint elements and graphical primitives. After modifying a script, the information can be saved, used to render a footprint, or exported. Script definitions are stored in library files with an extension of **.dsc**. See Using the Script Editor for how to create and modify scripts.

Using the Component Editor

The Electronics Desktop **Component Editor** supports the viewing, modification, and saving of component information for use in the development of circuit designs. You can create new components, edit existing components within a project file, and manage components within external libraries. After modifying a component, the information can be saved, used to define a design, or exported. Component definitions are stored in library files with an extension of **.aclb**. You can access component definitions using the **Component Editor**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To start the **Component Editor**, open a project and select **Tools** > **Edit Libraries** > **Components**. When the Edit Libraries window opens, use its controls to open the Edit Component window, then create a new component, or select an existing one, and click **Edit** to open the **Component Editor**.

Creating and Editing Components

Use the Edit Component window to

- specify or change the component name, description, associated bitmap, and default property values, including those necessary for netlisting, co-simulation, and **Solver-on-Demand** operation, as appropriate,
- specify or respecify a graphical symbol to represent the component in the schematic editor,
- specify or change the component terminal properties,
- specify or respecify a footprint to represent the component in the layout editor.

The changes you make when creating or editing a library component become part of the current project. The newly edited component is saved to the project, not back to the library. If the edited component is used in any schematics in the project, all default values of any modified parameters are updated immediately.

- To make a new or modified component available for use in other projects, save it to a component library (**.aclb**) file using the **Export** command in the Edit Libraries window.
- To update another project with your updated component(s), select its icon in the **Project Manager** window, then click **Project Tools** > **Update Definitions** on the **Tools** menu.

> **Note:** If you are creating a new component, it is useful to first identify and, if necessary, create the necessary dependencies before defining your component. See Component Creation Process in the introduction for further information.

Related Topics

Editing an Existing Component

Creating a New Component

Editing an Existing Component

To edit an existing component, do either of the following:

- From the **Project Manager** window, expand the **Definitions** /**Components** subfolder for the project that contains the component you want to edit. Double-click the entry for the component you want to edit, or right-click the entry and select **Edit Component**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Open the **Tools** > **Edit Libraries** > **Components** window, then locate and select the component that you want to edit. Click **Edit Component**, or double-click the selected entry.

The Edit Component window opens, displaying the definition of the selected component for editing.

Creating a New Component

To create a new component, do either of the following:

- Select **Tools** > **Edit Libraries** > **Components**, and click **Add Component** to open the **Edit Component** window.
- Open the **Tools** > **Edit Libraries** > **Components** window, then locate and select an existing component on which to base your new component. Click **Clone Component**. and a renamed copy of the selected component is added to the list. Double-click the entry for the new component, or click **Edit Component** to open the **Edit Component** window.

The Edit Component Window

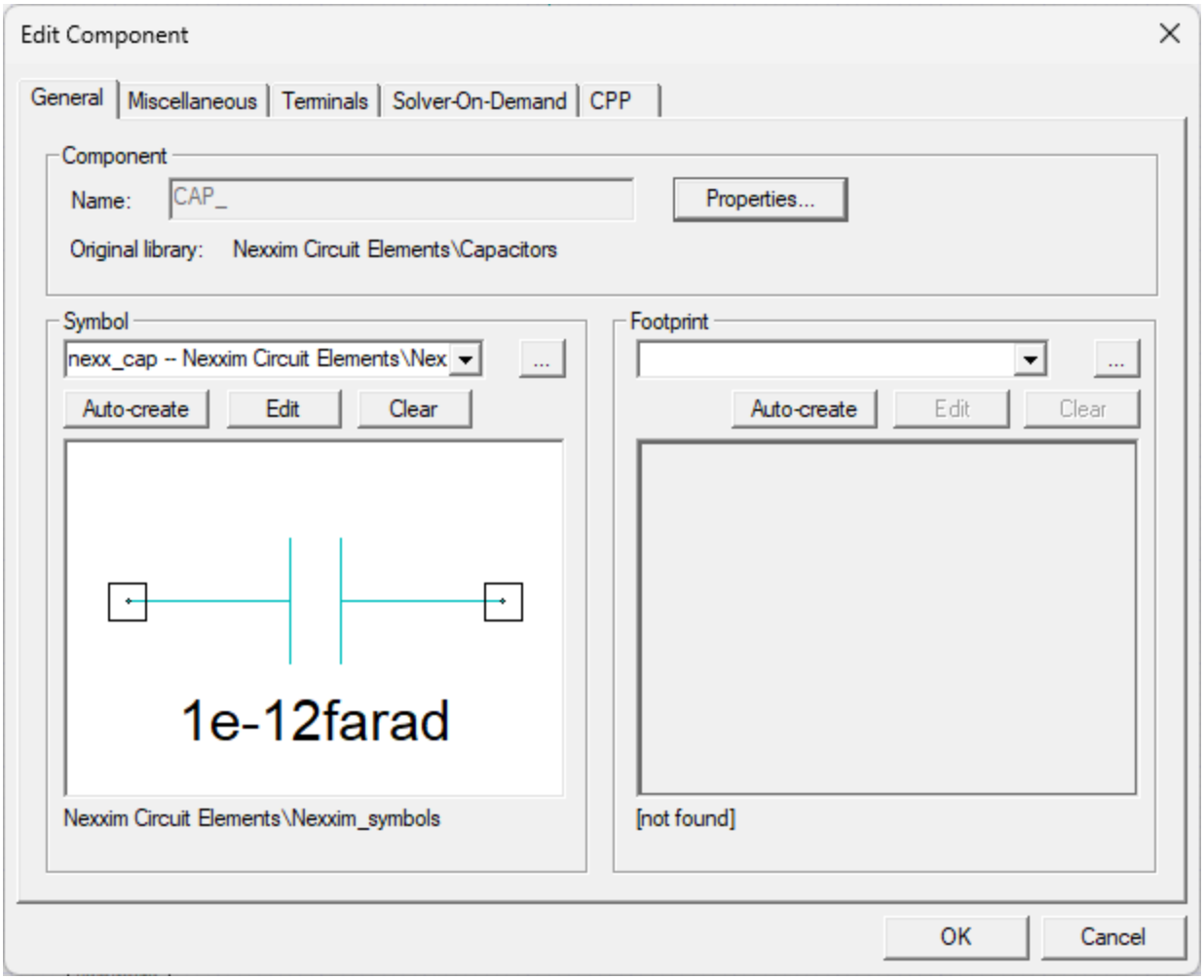
The **Edit Component** window is used to create new components, or edit existing components, in a project file. To show the **Edit Component** window, do either of the following:

- Double-click a component definition in the project tree, or double-click a component listed in the **Edit Library** window.
- Open a project and select **Tools** > **Edit Libraries** > **Components**. When the Edit Libraries window opens, use its controls to open the **Edit Component** window.

Components General Tab

Click the **General** tab to specify all parameter values for the component and assign symbols and footprints on their respective libraries. You can also start the symbol and footprint editors on the window, however, you must exit the component editor to complete the symbol or footprint edits. The symbol and footprint preview windows at the bottom of the window are automatically updated to reflect whatever changes you make to the symbols and footprints you modify.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Component Panel

The Component group box of the General Tab window displays the component Name. The Original Library name is displayed if found. To access and edit the component properties, click **Properties** next to the **Name** text window. In the Properties window, add, edit or remove default or local properties and values.

Symbol Panel

The Symbol group box of the General Tab window allows you to search by symbol name, and also displays a picture of the symbol. Choose any symbol which has an equal or greater number of pins than the component specified in the Component Name field. You can then choose to **Auto-create** the component.

The library of origin of the component is displayed beneath the component display in the **Symbol** area; if the component is **local to this Project**, that message is displayed. When you

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

click **Select** (**...**), the Select Definition window opens and allows you to choose a definition for the symbol from a list that is displayed. Click **Edit** to modify the symbol. A confirmation message is displayed with a reminder that the Component and Definition editors is closed (if open) and that all changes are saved if you choose to continue. Click **Clear** to clear all selections and displays in the Symbol area.

Footprint Panel

The Footprint group box of the General Tab window allows you to search by footprint name, and is used to display a picture of the footprint. Choose any footprint which has an equal or greater number of pins than the component specified in the Component Name field.

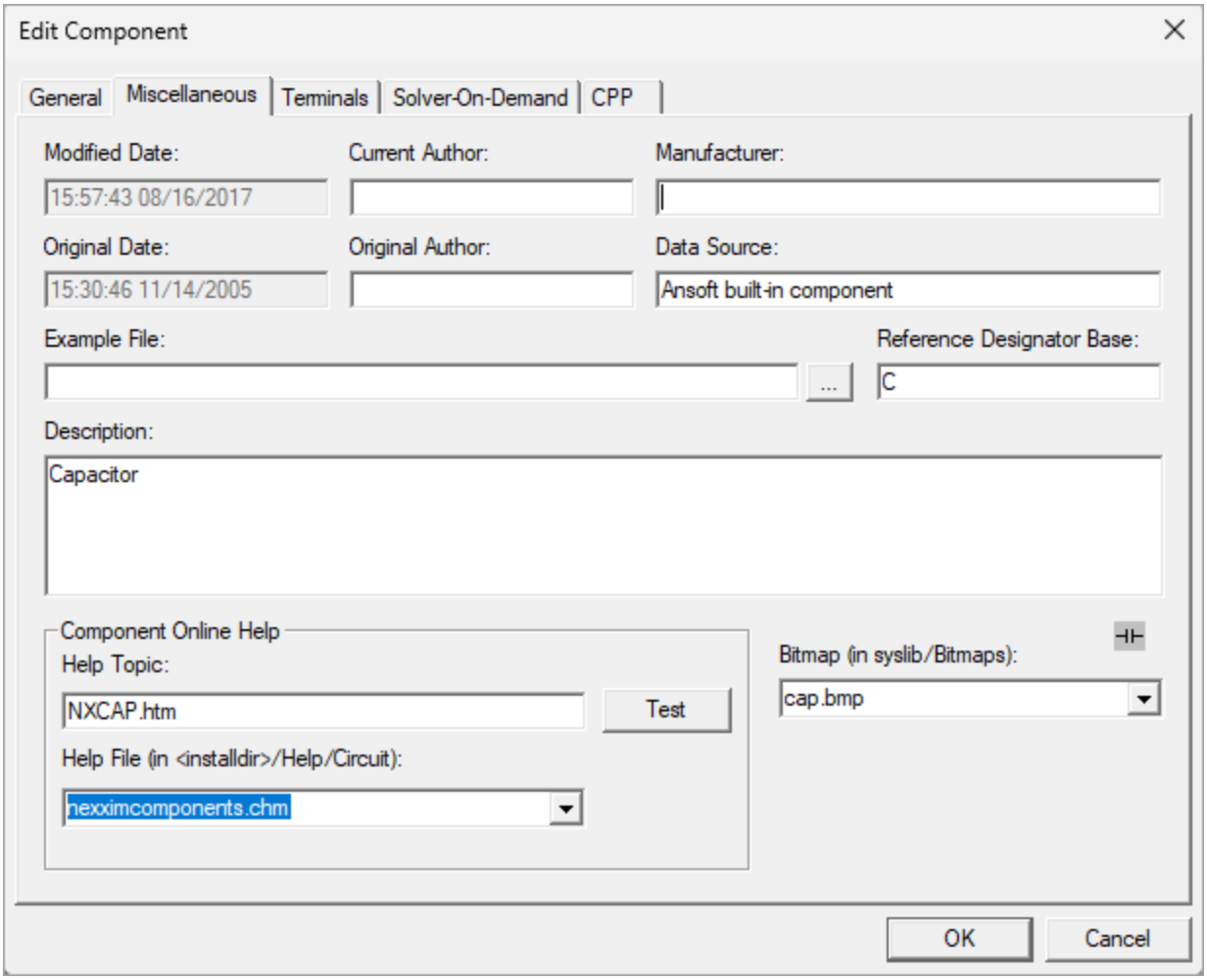
The library of origin of the footprint is displayed beneath the component display in the Footprint area; if the component is **local to this Project**, that message is displayed. When you click **Select**, (**...**) the Select Definition window opens and allows you to choose a definition for the footprint from a list that is displayed. Click **Edit** to modify the footprint. A confirmation message is displayed with a reminder that the Component and Definition editors is closed (if open) and that all changes are saved if choose to continue. Click **Clear** to clear all selections and displays in the Footprint area.

Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Components Miscellaneous Tab

Use the miscellaneous fields to enter component information such as the name, manufacturer, description, ref des, etc. You can also specify a bitmap for the **Project Tree** and an associated help description (if you use your own documentation). Help files must be located in **<Installation Directory>\Electronics Desktop\help**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



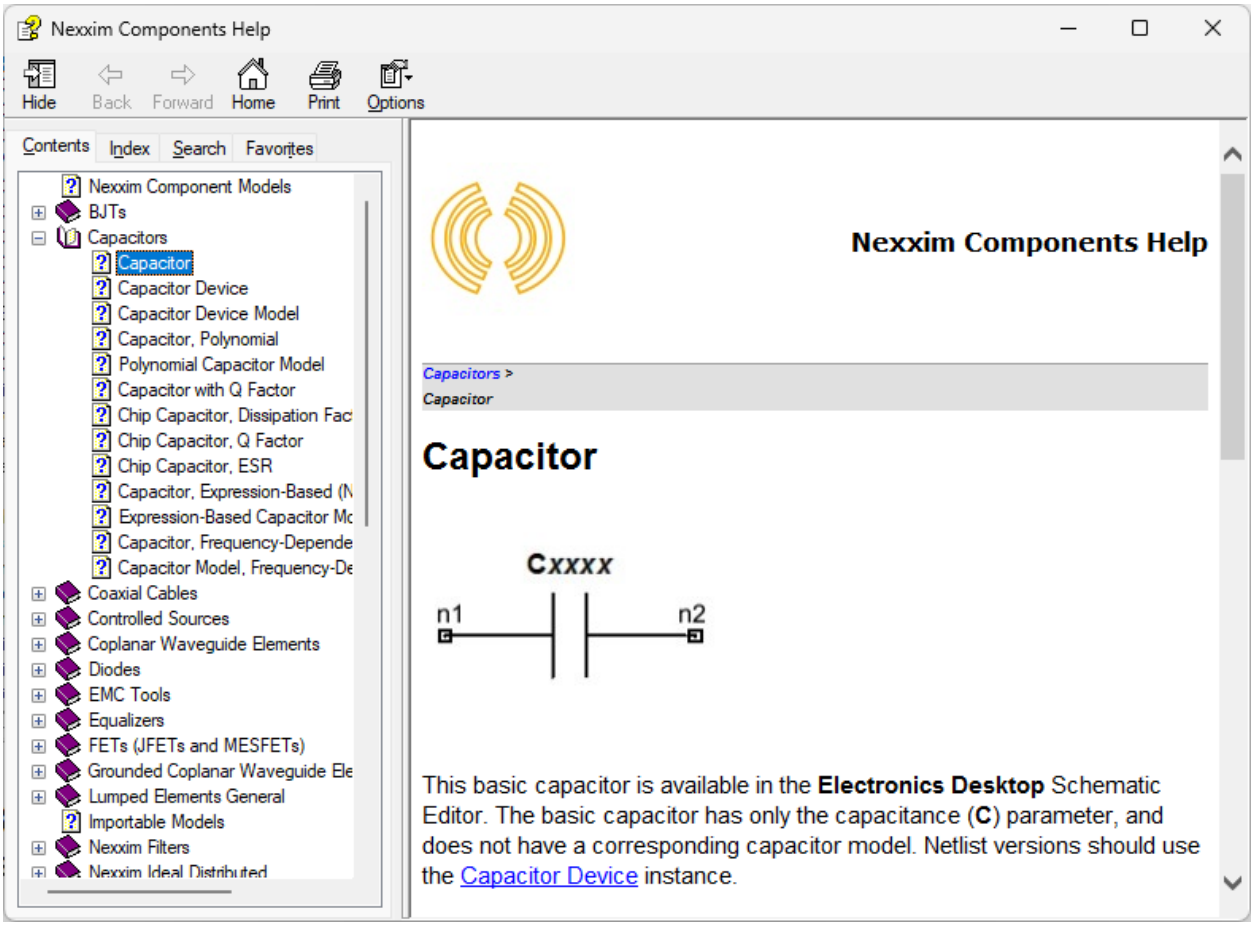
The top two rows of the **Miscellaneous** tab window displays the **Modified Date** and **Original Date** of the component, and allows you to specify the following:

- Current Author
- Manufacturer
- Original Author
- Data Source

You may also specify an **Example File** directory for the component, browse for the directory if necessary (...) and specify its **Reference Designator**, or enter a text **Description** for the component. The default **Bitmap** of the component is displayed at lower-right, and you may use the adjacent drop-down menu to specify a different bitmap file.

The **Component Help** area displays the default HTML help file associated with the component, and allows you to specify an alternative help file of your own. You may also **Test** the specified help file, which displays the component's HTML help page.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

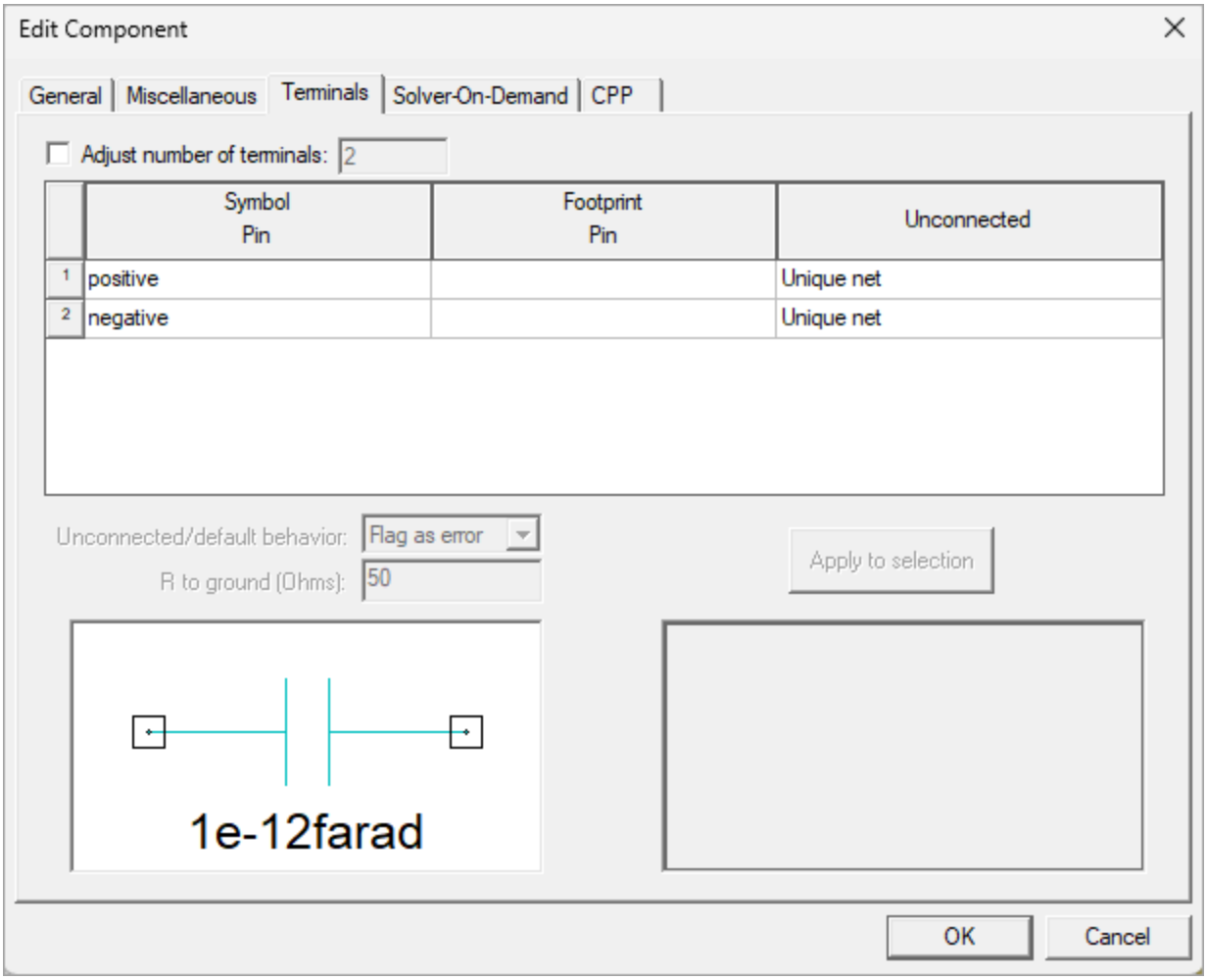


Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Components Terminals Tab

To line up component terminals in an envisioned order, Click the **Terminals** tab to associate each symbol pin with the proper terminal by first ordering the symbol pins, then selecting the corresponding symbol. This allows you to visually associate the symbol pins with the netlist properties for the simulation model. The symbol and footprint preview windows at the bottom of the window are automatically updated to reflect whatever changes you make to the symbols and footprints you modify.

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 controls are available:

In the **Terminals** display window, click one or more terminals directly, which then enables the following window options:

Unconnected/default behavior – Used to specify the response when a pin is not connected to a wire or port in a schematic. Choose between the following drop-down menu actions: **No action** (default), **Flag as error**, and **Grounded**. Click **Applytoselection** to apply your changes to the selected terminals.

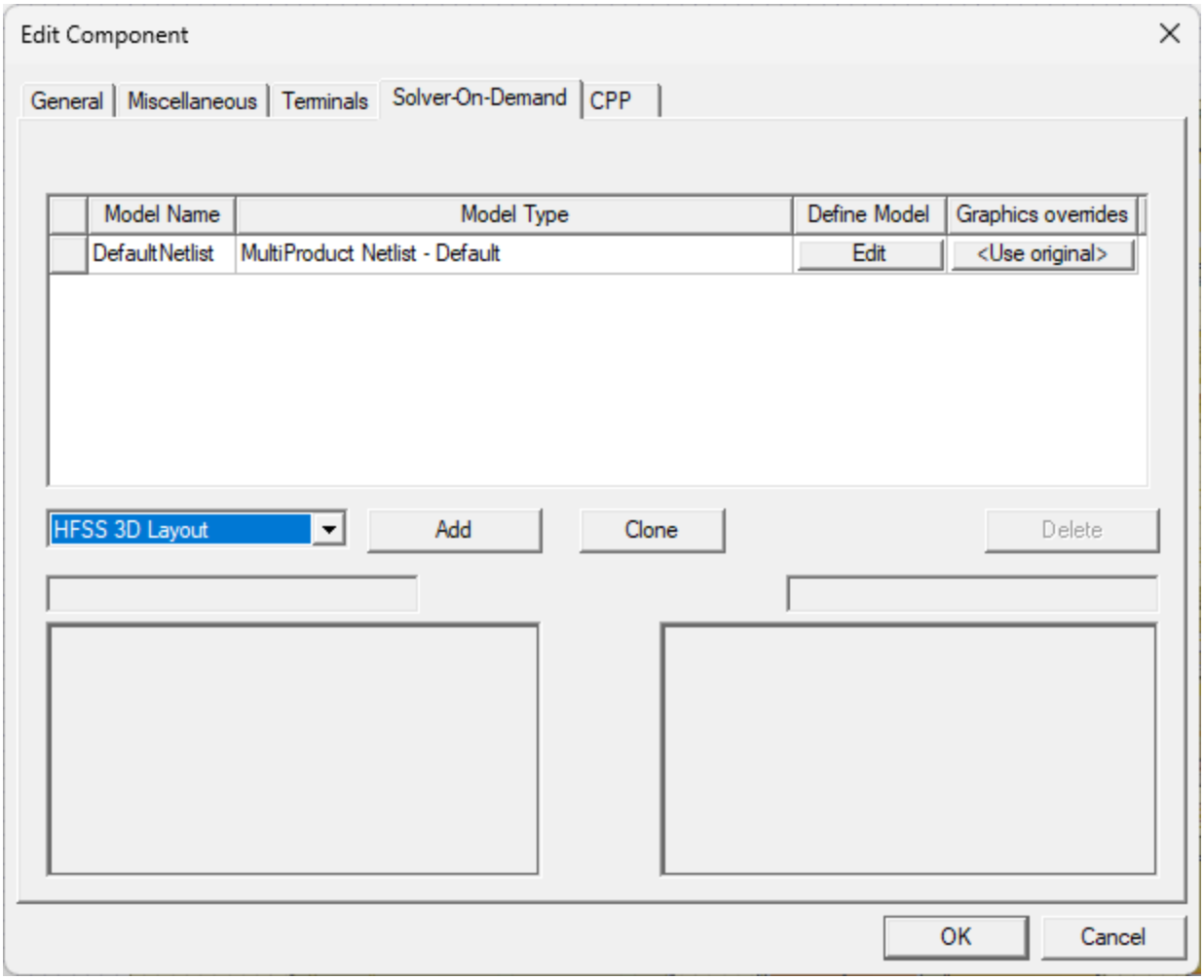
R to Ground – If you select **Grounded** for the **Unconnected/default** behavior, then specify the resistance to ground in the **R to Ground** entry window. Click **Applytoselection** to apply your changes to the selected terminals.

Click **OK** to save your changes and close the window, or click **Cancel** to close the window 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.

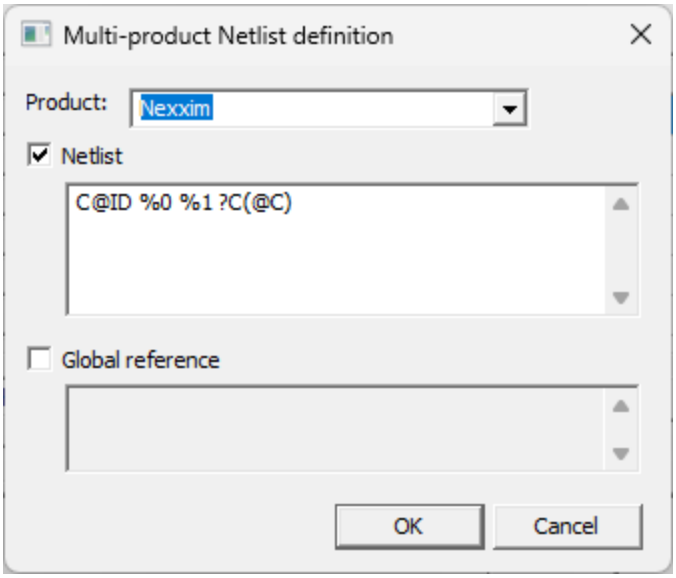
Components Solver On Demand Tab

Solver-on-Demand is a specialized routine to set up co-simulation using multiple products. When you first open the **Solver-on Demand** tab of the **Edit Component** window for a particular component, the only **Model Name** listed is the default multi-product type, **DefaultNetlist**.



DefaultNetlist is a default co-simulation definition type, provided automatically for each component, that allows you to define a different netlist for each Electronics Desktop product. Click the **Edit** button in the **DefaultNetlist** row to open the **Multi-product Netlist 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.



- Click the drop-down menu **Product** menu to choose which Electronics Desktop product netlist to define.
- Click **Netlist** to define a netlist.
- Click **Global reference** to define a global reference.
- Click **OK** to implement your changes, close the window, and return to the **Edit Component** window.

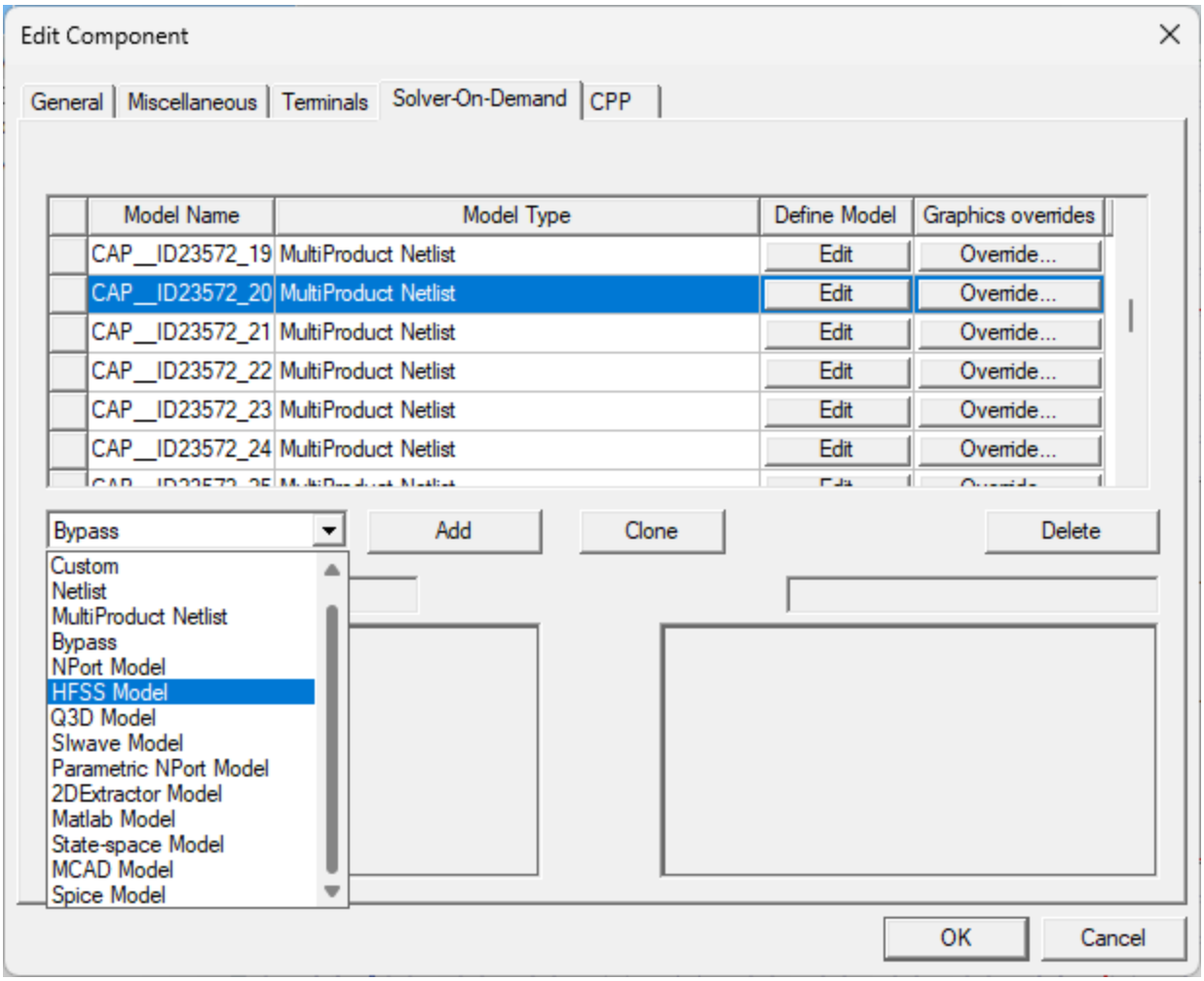
Multiple Cosimulation Models

Electronics Desktop allows for multiple representations of models that can be cosimulated. In the **Solver-on-Demand** tab of the **Edit Component** window, click in the drop-down menu to choose from a list of models to add to the list of **ModelNames**.

To add a model for Solver On Demand:

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 a selection on the model type drop-down menu.



2. Click **Add**. A new model of the selected type is added to the list. The symbol and footprint preview windows at the bottom of the window are automatically updated to reflect whatever changes you make to a component's symbol/footprint definitions.
3. Click in the **ModelName** field for the new model and type the name you want to use for this model (or accept the default in the **ModelName** field). Note that when you highlight a **Model Name**, its symbol and footprint (if defined) are automatically previewed in the windows at the bottom of the window. These preview windows are automatically updated to reflect whatever changes you make to the component's symbol/footprint definitions.
4. Click **Edit** in the **Define Model** field (or with a **Custom** model name) to open a **Cosim** or **Netlist** definition window that is specific to the model type.
5. Repeat steps 1 through 3 to create all the models you wish to use for this type of component.
6. Click **OK** when you have defined all the models for this component type.

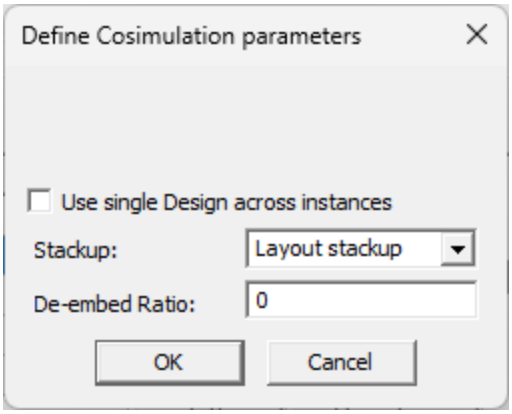
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Repeat the above procedure to specify models for all components that is simulated using **Solver-on-Demand**.

For a complete description of the Solver On Demand tab and how to define model types, see Cosimulation.

HFSS 3D Layout as Solver On Demand

HFSS 3D Layout is the available field solver that simulates from a layout, substrate, or footprint defined in the component itself. From the **Solver-on-Demand** tab of the **Edit Component** window, select **HFSS 3D Layout** from the drop-down menu, and click **Add** to open the co-simulation definition window.



- **Stackup** – Set to one of three values (Layout, Substrate, or Footprint) to determine which stackup definition are used to model the component when it is simulated by the selected solver. For more information see Stackup Definitions.

Solver-on-Demand with the HFSS 3D Layout tool requires the component possess a physical definition, in the form of a footprint. Many components in the Ansys Electronics Desktop® library have footprints defined so they can be simulated using HFSS 3D Layout as the **Solver-on-Demand**. When you define a component, it can be simulated using **Solver-on-Demand** as long as the component possesses a physical definition. For more information see Physical Definition.

- **Deembed Ratio** – Determines the manner in which excitation sign als are fed to the component via transmission lines during EM simulation. Short sections of transmission line are automatically added to the ports of the **Solver-on-Demand** component to allow discontinuities near the ports to decay in the EM simulation. The length of these transmission lines is equal to the Deembed Ratio times the width of the **Solver-on-Demand** port. That is, Length = Deembed Ratio * W (port connection). The solution data is automatically deembedded back to the initial **Solver-on-Demand** component geometry to remove the effects of the additional transmission lines.

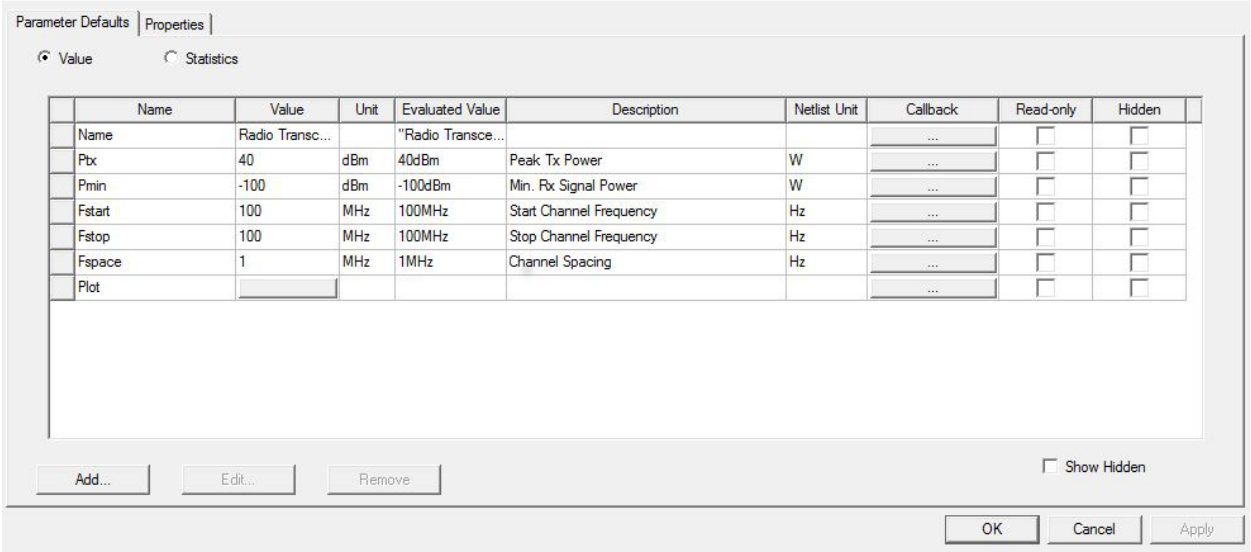
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Use single Design across instances** enables the same design to be used for all instances of the component type, eliminating the need to specify the **Solver-on-Demand** model individually.

Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Edit Component Properties Window

To open the **Properties** window for a selected component, double-click the component definition in the **Project Manager** window to open the **Edit Components** window. Then, in the **Edit Components** window, click **Properties** to open the window.



Parameter Defaults Tab

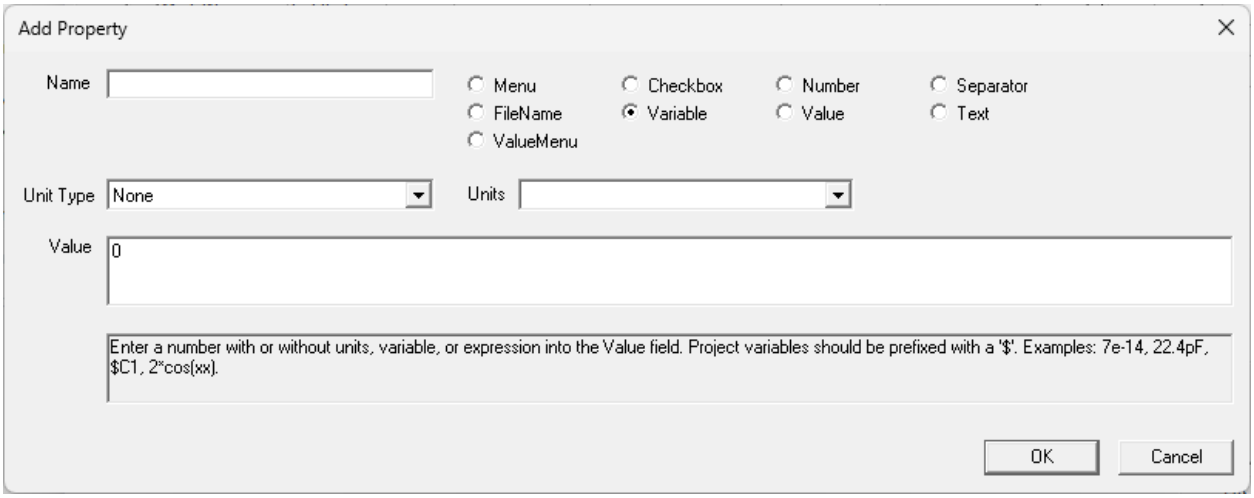
This tab displays and sets the default values for component parameters for general analysis (Value display) or statistical analysis (Statistics display).

When this option is selected, the **Parameter Defaults** tab lists, displays, and sets the default values for component parameters during general analysis.

Adding Parameters

Use **Add** to add a parameter to the definition. You are prompted to define the parameter name, value, and value type (text input, menu, check box, etc.). You may **not** add reserved system parameters to the definition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Removing Properties

To remove a property on a component, select the property and click **Remove**. The property is deleted on the component definition (model) and from all instances in the design.

> **Warning:**
>
> Removing a property from a component from an Electronics Desktop component library may produce undesirable results when the component is simulated.

Related Topics

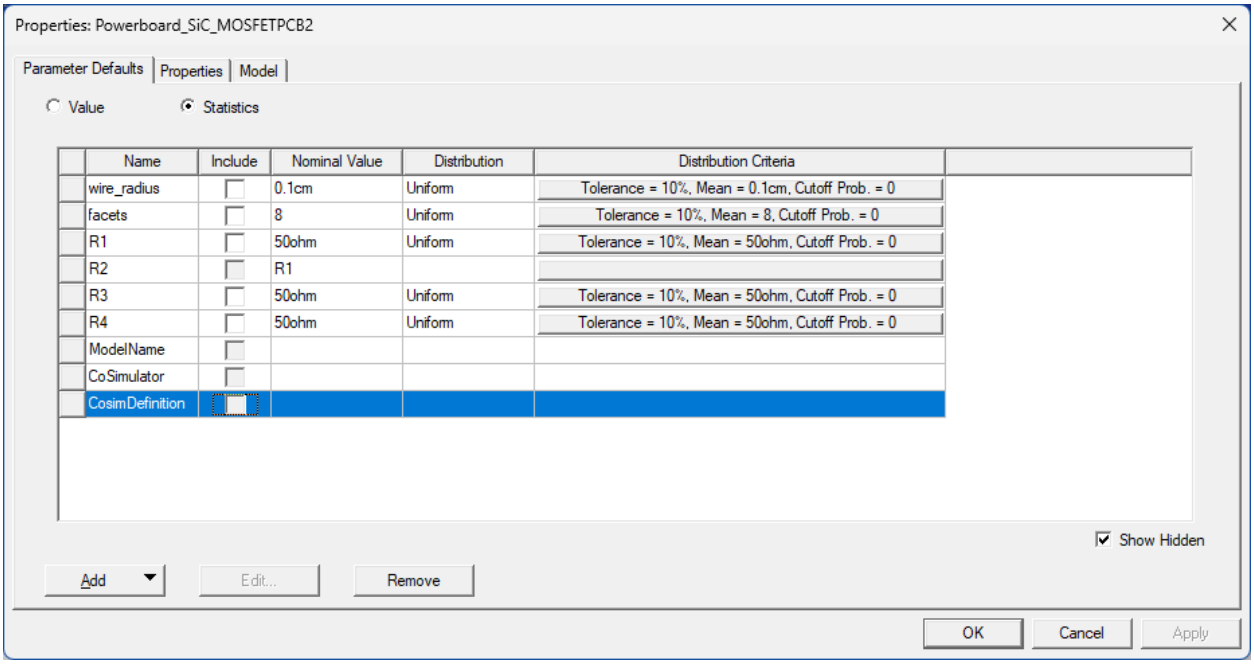
Parameter Statistics Display

Reserved Component Parameters

Parameter Statistics Display

When this option is selected, the **Parameter Defaults** tab lists, displays, and sets the default statistics for component parameters during general analysis.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



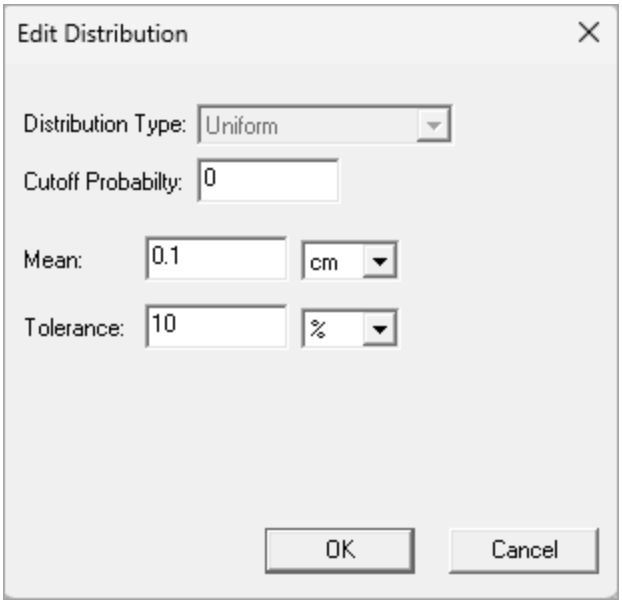
Name —Displays the names of component parameters.

Include —Controls whether or not the associated component parameter is varied during statistical analysis. Check **Include** for each parameter you want Electronics Desktop to vary during statistical analysis.

Distribution —Displays and controls whether the parameter value distribution is Uniform or Gaussian. To change the Distribution setting, click in the cell to open the options, then click the option you want.

Distribution Criteria —Click the display bar to open the **Edit Distribution** window. The **Distribution Type** drop-down menu displays whether the distribution is Uniform or Gaussian, and you can specify values for **Cutoff Probability**, **Mean**, and **Tolerance**. Units for **Mean** and **Tolerance** can be set using their adjacent dropdown 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 **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Reserved Component Parameters

When using **Add** to add a parameter to the component definition in the **Properties** window, you are prompted to define the parameter name, value, and value type (text input, menu, check box, etc.). You may **not** add the following reserved system parameters to the definition:

"Id"

"Status"

"AnalysisSetup"

"AnalysisResults"

"PartName"

"Info"

"NPortData"

"Refbase"

"NumParts"

"CompName"

"Description"

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

"Manufacturer"

"Datasource"

"Date"

"Symbol"

"Footprint"

"PinCount"

"Refdes"

Component CosimDefinition Property

A netlist is a textual representation of a circuit. The Electronics Desktop simulators extract a netlist on the schematic and perform the simulation on the netlist. The Nexxim simulator uses the Nexxim netlist format. Nexsys components within a Nexxim design use a legacy Ansys netlist format for components.

When an analysis is requested for a modified design, or when **Browse Netlist** is invoked, Electronics Desktop writes the corresponding netlist to the project results folder. The netlist conveys to the simulation engine the connectivity, parameter values, analysis specifications, and other characteristics of the design(s) under analysis. To support this process, every Electronics Desktop component must include a valid definition for the **CosimDefinition** property.

To view the **CosimDefinition** for a component:

1. Open its **Properties** window.
2. Select the **Component** tab.
3. Click **Edit** for **CosimDefinition**.

The **Multi-product Netlist definition** window opens, displaying the netlist string. Use the window to edit the string if appropriate.

Related Topics

Netlist String Syntax

Netlist String Syntax

The general form for a netlist string is:

<Simulator>Netlist=[netlist_string]

where

<Simulator> is Nexxim | Q3D | etc.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The netlist string may contain the following netlist property syntax:

%<terminal index, 0-based>

Which specifies the name of the net connected to terminal index. A syntax error results if the terminal index is not a positive number, within range. Example:

%0, %1

%_<internal node index, 0-based>

Create a unique net for an internal node. Used when a component has nodes that are not associated with symbol terminals and are not tied to a global node. Syntax error if node index is not a positive number, within range. Example:

%_0, %_1

@<propname> | @(<propname>)

Value of property named propname. Syntax error if propname is empty or contains spaces. Example:

@R, @IDSS

?<propname>(<expr1>)[:(<expr2>)]

If property named propname exists, substitute expr1, else expr2. Also, expr1 and expr2 may contain additional substitutions. Syntax error if propname is empty or contains spaces. Example:

?IDSS(IDSS = @IDSS):(IDSS = 0.05)

~<propname>(<expr1>)[:(<expr2>)]

If property named propname doesn't exist, substitute expr1, else expr2. Syntax error if propname is empty or contains spaces. Example:

~SUB(SUB = MS)

?(<propname>==<value>)(<expr1>)[:(<expr2>)]

?(<propname>!=<value>)(<expr1>)[:(<expr2>)]

~(<propname>==<value>)(<expr1>)[:(<expr2>)]

~(<propname>!=<value>)(<expr1>)[:(<expr2>)]

Same as "?" and "~" above except the first term is evaluated for equal to (==) or not equal to (!=) and if true, substitute expr1 else expr2. Example:

?(sim==fullwave)(NSUM=@NSUM):(F0=@F0)

***<propname>(<expr1>)**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If property named propname has changed from default definition value, then substitute expr1. Syntax error if propname is empty or contains spaces. Example:

*IDSS(IDSS = @IDSS)

&(<expr>)[^(pname1,pname2, . . .)]

Add all properties that have changed from default definition value, except those in the optional exclusion list. In expr, “$” can be used to represent the property name, and “#” the property value. Example:

&($=#)^(Model)

netlists all properties that changed on their default value except the Model property.

\n

A new line marker to inform the netlister to insert a new line. Example:

R@ID %0 %_0 @R \n C@ID %_0 %1 @C

****

The backslash is used as an escape character. The character following the backslash is not processed but added to the netlist string as is. To add a single “\”, use “\\”. Example:

R:@ID %0 %1 R=\{\2*@R\}

Defined Variables

- $SYSLIB is expanded to **<InstallationDirectory>/syslib** upon netlisting, where **<Installation Directory>** is the directory into which Electronics Desktop was installed during setup.
- $USERLIB is expanded to **<InstallationDirectory>/userlib** upon netlisting.
- $PERSONALLIB is expanded to **<ProjectDirectory>/PersonalLib** upon netlisting, where **<ProjectDirectory>** is the location you specified for your Electronics Desktop projects.

Global Reference String in Electronics Desktop

A global reference property processes its value the same way as the netlist string and places the result in the top-level (global) part of the circuit file.

<Simulator>GlobalRef=[string]

where

<Simulator> is Nexxim | Q3D | etc.

Additional Examples

The following additional examples help to demonstrate netlist string syntax.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NexximNetlist= CPLE@ID %0 %1 %2 %3 ?Z(Z=@Z) ?E(E=@E) ?F(F=@F) *A(A=@A)

“Z”, “E”, and “F” use the **"?"** conditional because the component definition values assigned to these parameters are different from those in the engine and should always be netlisted if they are present even if the user has not changed them. The “A” parameter uses the **"*"** conditional because its component definition value matches that of the engine and it needs to be netlisted if the user changes the instance value.

NexximNetlist= R@ID %0 %1 @R ?TC1(TC1=@TC1 ?TC2(tc2=@TC2)) ?TJ(TJ=@TJ) ?TNOM(TNOM=@TNOM)

In this example, TC1 is netlisted only if TC1 exists, and TC2 is netlisted only if TC1 and TC2 exist.

NexximNetlist= MSBENDO@ID %0 %1 w=@Length sub=@Substrate

“w” and “sub” is netlisted.

NexximNetlist= MOSFET@ID %0 %1 %2 %3 MODEL=@Model &($=#)^(Model)

Model is netlisted. The “&($=#)^(Model)” argument designates that all parameters should be netlisted as “name=value”, except for “Model”.

NexximNetlist = D@ID %0 %1 1N914

NexximGlobalRef = .LIB $SYSLIB/Vendor/D.lib

which may netlist to:

D22 net_3 net_4 1N914

< rest of design netlists ...>

.LIB $SYSLIB/Vendor/D.lib

NexximNetlist= R@ID %0 %1 R=\{\2*@R\}

If ID is 1 and R is 100, the netlist is:

R1 net_0 net_1 R={2*100}

Using the Symbol Editor

A symbol is a set of information that defines the graphical representation and electrical connectivity of a component. Symbol definitions are stored in library files with an extension of **.aslb**.

You can access material definitions using the **Symbol Editor**. The Electronics Desktop **Symbol Editor** allows you to view, modify, and save symbol information for components and layout

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

primitives. After modifying a symbol, the information can either be saved, used to define a design, or exported.

When a symbol is used in a project, it is copied to the project. Thus new projects do not list any symbols in the Project Manager. If a change is made to the symbol, it is actually made to the project's copy of the symbol. So each project has its own set of symbols based on AEDT's symbol library. When you open an existing project, the Project Manager displays the symbols in use in the project.

To start the **Symbol Editor**, open a project and select **Tools** > **Edit Libraries** > **Symbols**. When the Edit Libraries window opens, use its controls to open the **Symbol Editor**.

Creating a New Symbol

1. Click **Tools** > **Edit Libraries** > **Symbols**.
2. From the **Symbols** tab of the **Edit Libraries** window, either:
   - Click **Add Symbol** to open the **Get Name** window.
   - Select an existing symbol to base your new symbol and click **Clone Symbol(s)** to open the **Get Name** window.
3. Type a name for the new symbol into the **Enter the name for this new Symbol** field and click **OK**.
4. Then double-click the entry for the new symbol or click **Edit Symbol** to open the symbol editor.
5. Use the symbol editor to:
   - Draw graphical primitives, such as rectangles, circles, and arcs, using options on the Draw menu.
   - Add pins for electrical connections using the Pin option on the **Draw** menu.
   - Modify pin properties in the Pin List dialog box.
   - Add labels. To include component properties or other variables in a label, use the Property Display Setup option on the Symbol menu. For unchanging text, use the text option on the Draw menu. Unusual labels are most likely represent a component property.
   - Update the current project with the new or revised symbol definition using the Update Project option on the **Symbol** menu

   You can also export a symbol to a symbol library (**.aslb**) file for use in other projects. For information on how to do this, see the Edit Libraries.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing an Existing Symbol

When a symbol is used in a project, it is copied to the project from a global library. Any changes made to the symbol are made to the project's copy of the symbol and only affect the symbol in the project and not in any other project. To edit an existing symbol, either:

- From the **Project Manager** window, open the project that contains the symbol you want and open **Definitions** > **Symbols**. Either double-click the symbol you want to edit or right-click the symbol and select **Edit Symbol**.
- Open **Tools** > **Edit Libraries** > **Symbols**. From the **Symbols** tab of the **Edit Libraries** window, either double-click the symbol that you want to edit or right-click the symbol and choose **Edit Symbol**.

The symbol editor opens with the selected symbol ready for editing.

Use the symbol editor to:

- Draw graphical primitives, such as rectangles, circles, and arcs, using options on the Draw menu.
- Add pins for electrical connections using the Pin option on the **Draw** menu.
- Modify pin properties in the Pin List dialog box.
- Add labels. To include component properties or other variables in a label, use the Property Display Setup option on the Symbol menu. For unchanging text, use the text option on the Draw menu. Unusual labels are most likely represent a component property.
- Update the current project with the new or revised symbol definition using the Update Project option on the **Symbol** menu

You can also export a symbol to a symbol library (**.aslb**) file for use in other projects. For information on how to do this, see the Edit Libraries.

Symbol Editor Operations

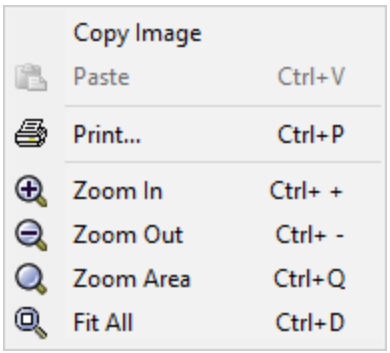
The symbol editor opens when you begin to create or modify a symbol. Like the **Schematic Editor**, the symbol editor is a graphical tool that keeps the placement of symbol elements uniform by snapping them to points on a grid. The window controls are the same as those in the schematic editor: press the arrow keys and **Page Up** and **Page Down** to pan, or **Ctrl** +**D** to fit the view.

With the symbol editor, add, modify, and manipulate graphical primitives, text labels, pins, and property displays. You can also adjust the resolution, color, and visibility of the editor grid. When you're done, update the current project with any changes you've made to the current symbol. Options related to these operations are available on the Symbol menu and the Symbol Draw menu.

Symbol Editor Shortcut Menus

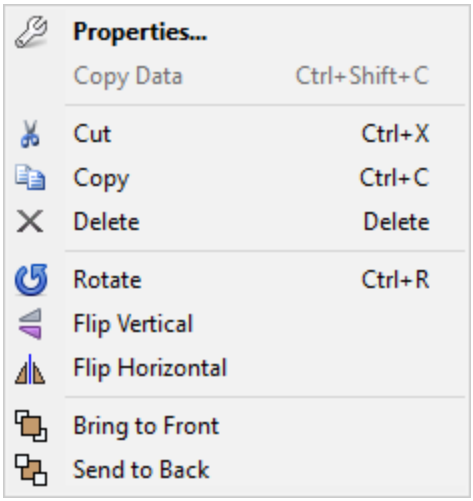
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you right-click in the symbol editor grid, this shortcut menu appears:



These options are a subset of those available on the **View** menu.

When you right-click an object in the symbol editor, this shortcut menu appears:



Click **Properties** to view the properties of the selected object. The other options are a combination of entries on the **Edit**, **View**, and **Draw** menus.

Symbol Draw Menu

The symbol editor **Draw** menu presents options for drawing graphical primitives and text, and manipulating selected objects. The **Draw** menu options include

- Arc
- Circle
- Line
- Polygon

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Rectangle
- Text
- Image
- Ellipse
- Curve
- Pin
- Rotate
- Align Horizontal
- Align Vertical
- Flip Vertical
- Flip Horizontal
- Bring to Front
- Send to Back

Arc

This option, also available on the **Symbol Draw** toolbar , initiates creation of an arc. To draw an arc, click the symbol editor grid at the two points that determines the arc's ends, then drag the arc to the appropriate radius.

Once you have created an arc, adjust its radius or move its endpoints by clicking the arc to select it, and dragging the appropriate handle.

Circle

This option, also available on the **Symbol Draw** toolbar , initiates creation of a circle. To draw a circle, click the symbol editor grid to select a center point, then drag the circle to the appropriate diameter.

Once you have created a circle, adjust its diameter in either of two ways:

- Click the circle to select it, then drag one of its handles.
- Double-click the circle to open its **Properties** window, type a new value for the **Radius** parameter, then click **OK**.

By default, a new circle is hollow. You can fill a circle with solid color or parallel lines in one of several styles as follows:

a. With the circle's properties displayed, click in the **Value** cell for the **FillStyle** property.
b. Select the appropriate fill style on the list.
c. 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.

Line

This option, also available on the **Symbol Draw** toolbar , initiates creation of a polyline with one or more segments. To draw a line:

1. Click the symbol editor grid where you want the line to start, then click at one or more points to continue the line.
2. To complete the line, do one of the following after defining its final segment:

- Press the **Spacebar**.
- Right-click, then click **Finish**.

After you have completed a line, change the endpoints of its segments by doing one of the following:

- Select the line and drag the appropriate handle(s).
- Double-click the line to open its **Properties** window, type new position values (in the form X: Y) for the appropriate vertices, and click **OK**.

Alternately, change the endpoints of a selected line by clicking **Layout** > **Line Styles** to open the **Line Styles** window, then click in the Cap Type field to change the cap or line-ending type (Flat, Extended, or Round).

Because a line has nonzero width, its CapType determines how its ends behave:

- **Extended** – Extends the line by its own width past the endpoint you click.
- **Flat** - Cuts the line perpendicularly at the endpoint you click.
- **Round** - Ends the line in a semicircle.

The existing technology files define line/trace styles, including a CapType (typically Flat) for each style. If you select None as the technology type, you'll automatically get a palette of four styles with a variety of widths, BendTypes and CapTypes.

Polygon

This option, also available on the **Symbol Draw** toolbar , initiates creation of a polygon. To create a polygon:

1. Click the symbol editor grid to specify the position of one vertex, then click wherever you want to place additional vertices.
2. To complete a polygon, specify the position of its final vertex, then do either of the following:

- Press the **Spacebar**.
- Right-click, then click **Finish**.

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 drawn a rectangle, Edit its vertex positions and other properties, including its fill style, as follows:

- Click the rectangle, then edit its properties in the **Properties** window.
- Double-click the rectangle, then edit its properties in the **Properties** window.

By default, a new polygon is hollow. You can fill a polygon with solid color or parallel lines in one of several styles as follows:

a. With the polygon's properties displayed, click in the **Value** cell for the **FillStyle** property.
b. Select the appropriate fill style on the list.
c. Click **OK**.

Rectangle

This option, also available on the **Symbol Draw** toolbar , initiates creation of a rectangle. To create a rectangle, click the editor grid to specify the position of one corner, then drag the rectangle to the appropriate size.

Once you have drawn a rectangle, Edit its height and width, the position of its center, and its angle (its rotation, in degrees, relative the handles of its bounding box) as follows:

- Click the rectangle, then edit its properties in the **Properties** window.
- Double-click the rectangle, then edit its properties in the **Properties** window.

By default, a new rectangle is hollow. You can fill a rectangle with solid color or parallel lines in one of several styles as follows:

1. With the rectangle's properties displayed, click in the **Value** cell for the **FillStyle** property.
2. Select the appropriate fill style on the list.
3. Click **OK**.

Text

This option, also available on the **Symbol Draw** toolbar , adds to the symbol an editable "Default text" label in 12-point Arial. To edit the default text string immediately after placement, type the text you want, then press **Enter** or click elsewhere in the editor grid.

To change just the text of an existing label:

1. Click the label.
2. Click the label again to open its text for editing.
3. Type the appropriate text.
4. Press **Enter** or click elsewhere in the editor grid.

To change other properties of a label, including its color, font or size:

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 the label and view its properties in the **Properties** window, or double-click the label and view its properties in the **Properties** window.
2. Click in the **Value** cell for the property you want to modify.
3. Modify the value.
4. Click **OK**, or click in another **Value** cell to commit the change and keep editing values.

Image

This option, also available on the **Symbol Draw** toolbar , adds an image to the symbol. To add an image, select an image file in the **Select Image** window and click **Open**. Click the editor grid to place the image, then drag the mouse until the image is the appropriate size, and click the mouse again.

Once you have placed an image, Edit its height and width, the position of its center, and its angle (its rotation, in degrees, relative the handles of its bounding box) as follows:

- Click the image to select it, then drag one of its handles.
- Click the image and edit its properties in the **Properties** window.
- Double-click the image and edit its properties in the **Properties** window.

To flip the image horizontally, select the image and in the **Properties** window, click **Mirrored**.

By default, a new image is not linked its original file. To ensure that the image is updated when changes are made to the file, select the image and in the **Properties** window, click **Link to File**.

A new image does not have a border. Add one as follows:

1. With the image's properties displayed in the **Properties** window, click the **Value** cell for the **DisplayBorder** property.
2. Set a **BorderWidth**.
3. Click the **Value** cell for the **BorderColor** property to add a color in the **Color** window.
4. Click **OK** to close the **Color** window.

Ellipse

This option, also available on the **Symbol Draw** toolbar , initiates creation of an ellipse. To draw a ellipse, click the editor grid to specify the top and left most position of the shape , then drag the ellipse to the appropriate size.

Once you have drawn an ellipse, edit its height and width, the position of its center, and its angle (its rotation in degrees relative the handles of its bounding box) as follows:

- Click the ellipse to select it , then drag one of its handles.
- Click the ellipse and edit its properties in the **Properties** window.
- Double-click the ellipse and edit its properties 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.

By default, a new ellipse is hollow. You can fill an ellipse with solid color or parallel lines in one of several styles as follows:

1. With the ellipse's properties displayed, click in the **Value** cell for the **FillStyle** property.
2. Select the appropriate fill style on the list.
3. Click **OK**.

Curve

This option, also available on the **Symbol Draw** toolbar , initiates creation of a polycurve with one or more segments. To draw a curve:

1. Click the symbol editor grid where you want the curve to start.
2. Clicke or more points to continue the curve.
3. To complete the curve, mark its final segment and do one of the following:
    - double-click.
    - Press the **Spacebar**.
    - Right-click and click **Finish**.

After you have completed a curve, change the endpoints of its segments by doing one of the following:

- Select the curve and drag the appropriate handle(s).
- Double-click the curve to open its **Properties** window, type new position values (in the form X: Y) for the appropriate vertices, and click **OK**.

Pin

This option, also available on the **Symbol Draw** toolbar, initiates placement of a symbol pin with a default stem length of 10. You can rotate a pin once you've started to place it by iteratively pressing R until the pin is oriented to your liking. Click in the editor grid to finish placing the pin.

Rotate

This option, also available on the **Symbol Draw** menu and by pressing **Ctrl**+**R** on the keyboard, rotates a selected object or group of objects 90° to the left.

Align Horizontal

This option horizontally aligns the uppermost edges of the members of a group of selected objects with the uppermost edge of the first-selected object. To align multiple objects horizontally:

1. Press **Ctrl**, then click the object with which you want to align the others.
2. Still pressing **Ctrl**, click the additional objects in turn to add them to 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.

Note that the first-selected object is highlighted in red, and that the subsequently selected objects are highlighted in dark red.

3. From the **Draw** menu, select **Align Horizontal**.

Align Vertical

This option vertically aligns the leftmost edges of the members of a group of selected objects with the leftmost edge of the first-selected object. To align multiple objects vertically:

1. Press Ctrl, then click the object with which you to align the others.
2. Still pressing Ctrl, click the additional objects in turn to add them to the selection.

   Note that the first-selected object is highlighted in red, and that the subsequently selected objects are highlighted in dark red.

3. From the **Draw** menu, select **Align Vertical**.

Flip Vertical

This option, also available on the **Symbol Draw** toolbar, flips a selected object about the X axis.

Flip Horizontal

This option, also available on the **Symbol Draw** toolbar, flips a selected object about the Y axis.

Bring to Front

This option, also available on the **Symbol Draw** toolbar, places a selected object in front of all other elements in the editor, i.e., closest to the user.

Send to Back

This option, also available on the **Symbol Draw** toolbar, places a selected object behind all other elements in the editor, i.e., furthest on the user.

Symbol Menu

The Symbol menu lists options for symbol naming, property displays, and file operations. Its entries include the following:

Update Project

This option updates the current project with changes made in the current symbol.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note: Update Project** updates your symbol changes to the current project in memory and does not save your changes to disk. To save your changes to disk, you must also click **Save** on the **File** menu. If you like the changes you have made in a symbol to be available for use in other projects, you must export the symbol to a library as described in the Edit Libraries topic.

Set Name

This option opens a **Properties** window to rename the current symbol.

Property Display Setup

This option opens the Properties Display so add or modify display of the names and/or values of properties associated with the component represented by the current symbol.

Pin List

This option gives you access to settings related to pin properties by opening the Pin List window.

Grid Setup

This option gives you access to settings related to the symbol editor grid resolution, color, and visibility by opening the symbol editor Grid Setup window.

Import File

The Import File command allows you to import an existing SVG-formatted symbol into the symbol editor.

Export File

This option saves the current symbol as a Microsoft Enhanced Meta File (**.emf**) graphic.

Edit Component

This option launches the Edit Component window to modify the component associated with the active symbol.

Normalize Symbol on Save

Allows you to normalize the symbol you are editing when you save after making modifications.

List

Opens the **Design List** window, which lists the objects in the layout. Each column may be sorted by clicking on the heading. One or more object rows can be selected and operated by clicking **Delete** and **Properties**.

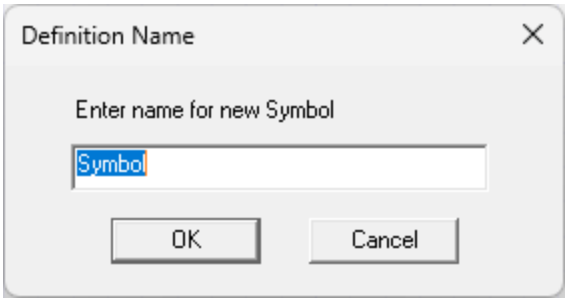
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import File (Symbol)

The **Symbol** > **Import File** command allows you to import an existing SVG-formatted symbol into the symbol editor. The SVG (Scalable Vector Graphics) format is an open, xml-based web standard graphics format supported by many third-party tools.

To import an SVG (**.svg**) file when creating a new symbol definition do the following:

1. From the **Project Manager** window, right-click **Definitions** > **Symbols** in the appropriate project and select **Add Definition**.
2. In the **Definition Name** window, enter a name for the new symbol and click **OK** to open a new blank **Symbol Editor** window.



3. From the main menu bar, select **Symbol** > **Import File** and choose an existing **.svg** symbol file to import it into the symbol editor. A window opens advising that importing the **.svg** file removes any existing symbol graphics in the editor, and asking for confirmation to continue the import.

> **Note:**
>
> - There is no “Undo” for the import operation. Consequently if you make a mistake on import, you must close the project without saving, then reopen it to recover.
> - Currently **.svg** symbols can be imported into any symbol editor window - including those containing pre-existing component symbols in which case the current symbol in the editor is replaced by the imported symbol, so ensure that you import the symbol in the appropriate place.

4. Click **OK** to import the **.svg** symbol for further editing, or **Cancel** to abort the operation. Once the new symbol has been created, attach it to a component via the Property Display Setup command.

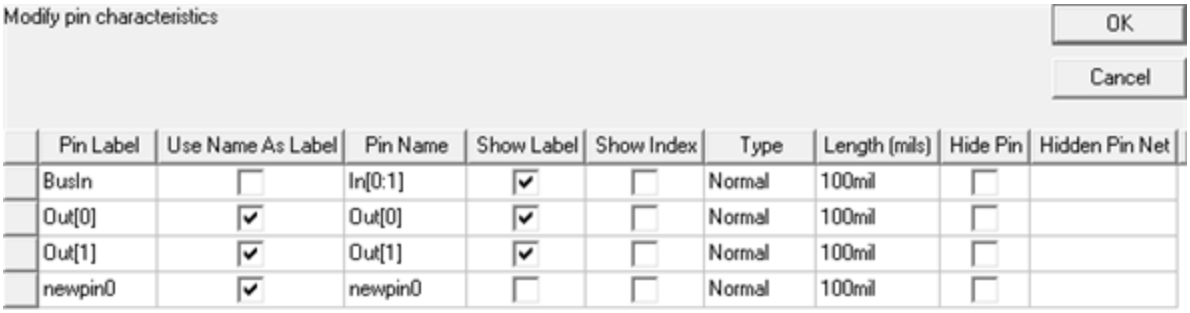
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Pin Properties

To edit a property of a symbol pin, open the symbol editor's **Pin List** window by doing either of the following:

- In the **Symbol** menu, select **Pin List**.
- From the **Symbol** toolbar, click the **Pin list** icon .

The **Pin List** window displays and sets values for the properties of symbol pins.



Pin properties include:

- **Pin Label** – provides a field to add a user-defined label for the chosen pin on a component symbol without having to change the actual **Pin Name** associated with the chosen pin. Pin labels for pins in a component should be unique.

  To specify or change the label of a pin, click in the **Pin Label** cell for that pin, type the name, and do either of the following:

  - Press **Enter**.
  - Click in another cell.

  The **Pin Label** can be displayed for the chosen pin by unchecking **Use Name As Label**.

- **Use Name As Label** – controls whether the **Pin Name** or **Pin Label** is displayed for the chosen pin when **Show Label** is enabled. For a new pin, **Use Name As Label** is checked by default.
- **Pin Name** – sets the actual pin name. To specify or change the name of a pin, click in the **Pin Name** cell for that pin, type the name, and do either of the following:
  - Press **Enter**.
  - Click in another cell.
- **Show Label** – controls visibility of the either the **Pin Name** or **Pin Label** text for the pin (as determined by **Use Name As Label**). For a new pin, it is unchecked 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.

- **Show Index** – controls visibility of the pin number in the schematic editor. For a new pin, it is unchecked by default.
- **Type** – drop-down menu sets the pin type. Options include **Normal** (the default), **ANSI In**, **ANSI Out**, and **Zero Length**. To change a pin's type, click in its **Type** cell, then click the appropriate option.
- **Length (mils)** – sets the length of the pin stem (the graphical line associated with the pin port symbol). By default, new pins have a pin stem length of 100 mils. To change the length of a pin, click in its **Length** cell, type a new value, and do either of the following:
  - Press **Enter**.
  - Click in another cell.
- **Hide Pin** – controls whether or not a pin is visible in the schematic editor. For new pins, it is unchecked by default.
- **Hidden Pin Net** – editable when **Hide Pin** is checked, specifies the net (circuit node) to which a pin is connected if it is hidden. To specify a net, click in the **Hidden Pin Net** cell, type a net name, then do either of the following:
  - Press **Enter**.
  - Click in another cell.

Edit Symbol Pin Locations

To edit the location of a symbol pin, open the **Edit Symbol Pins** window by doing one of the following:

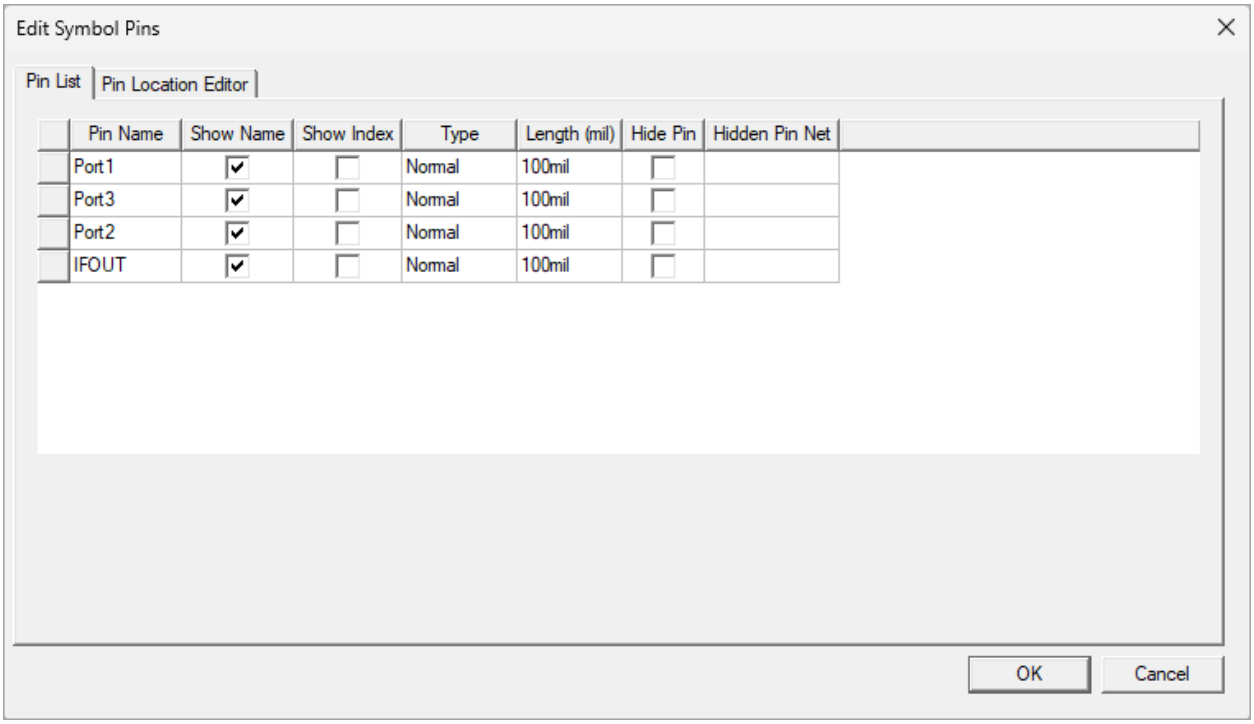
- In the Symbol Editor **Symbol** menu, select **Edit Pins**.
- In the component instance shortcut menu, select **Edit Symbol Pins**.

The window has three tabs: **Pin List**, **Pin Location Editor**, and **Bus Pin Editor**. Pin Location Editor and Bus Pin Editor appear only for symbols that are drawn as a rectangle with pins.

Pin List

The **Pin List** window displays and sets values for the properties of symbol pins.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The available pin properties include

- **Name** – This editable cell displays and sets the pin name. To specify or change the name of a pin, click in the **Name** cell for that pin, type the name, then press **Enter** or click in another cell.
- **ShowLabel** – This check box controls visibility of the pin name in the schematic editor. For a new pin, it is unchecked by default.
- **ShowIndex** – This check box controls visibility of the pin number in the schematic editor. For a new pin, it is unchecked by default.
- **Type** – This drop-down menu sets the pin type, the options for which include **Normal** (the default), **ANSI In, ANSI Out,** and **Zero Length.** To change a pin's type, click in its **Type** cell, then click the appropriate option.
- **Length** – This editable cell sets the length of the pin stem (the graphical line associated with the pin port symbol). By default, new pins have a pin stem length of 10. To change the length of a pin, click in its **Length** cell, type a new value, then press **Enter** or click in another cell.
- **HidePin** – This check box controls whether or not a pin is visible in the schematic editor. For new pins, it is unchecked by default.
- **HiddenPinNet** – This cell, editable when **Hide Pin** is checked, specifies the net (the circuit node) to which a pin is connected if it is hidden. To specify a net, click in the **Hidden Pin Net** cell, type a net name, then press **Enter** or click in another cell.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

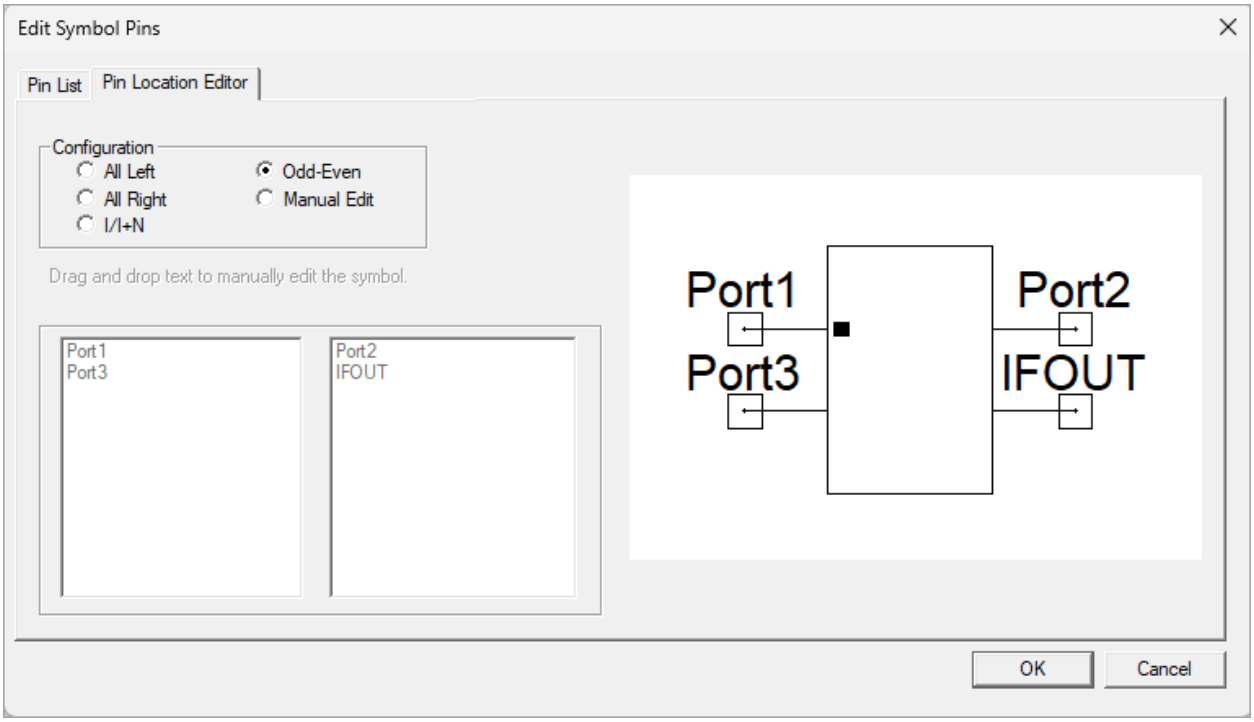
Changing Pin Properties

To change a pin property, either

- Click the cell that displays the pin property value that you want to change and modify the property value appropriate to its type as described above.
- To commit the new value and continue editing pin properties, click another property value.

Pin Location Editor

The **Pin Location Editor** allows you to change the locations of the symbol pins to the left or right side according to the settings chosen in the window.



Selecting a new option on the **Reference** drop-down menu changes the symbol displayed to show or hide additional reference pins. The **Reference** list is not shown in the window if the symbol does not support reference pins. The options are

- **Implied reference to ground**: the global ground is the reference port for each N port connection. Its voltage is always 0V.
- **Common reference port**: an additional reference pin is added that is a common reference for the definition of port voltages. This reference can have any voltage and corresponds to the net for that extra pin.
- **Add individual hidden reference pin per port**: an additional reference pin is added for each port. These reference pins are hidden, set to 0V by default, and can each have their

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

own reference voltage. You can connect each reference pin to its own net. Select this option if you do not want to see the reference pins.

- **Add individual reference pin per port**: an additional reference pin is added for each port. Every reference pin can have its own reference voltage and be connected to its own net.

> **Note:**
>
> See Reference Nodes on S-Parameter Elements in the S-parameter Technical Notes for how the Circuit solver handles reference node configurations.

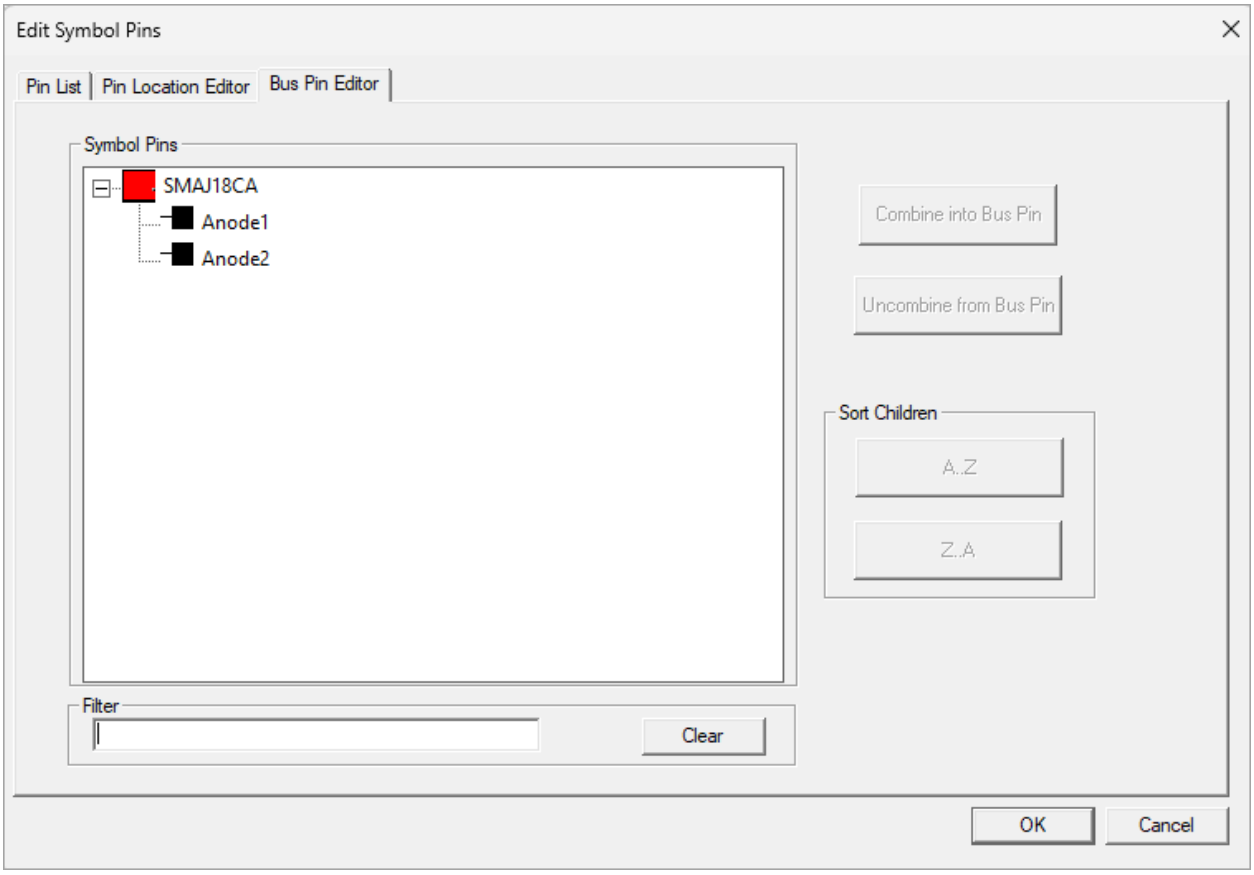
Clicke of the five **PinDirection** options to change the symbol shown to a fixed configuration.

- **All Left** puts all pins on the left side
- **All Right** puts all pins on the right side
- **I/I+N** puts the first half of the pins in order on the left side and the remaining pins on the right side
- **Odd-Even** puts every other pin on the left side and the remaining pins on the right side
- **Manual Edit** enables the two list boxes in the bottom left quadrant. When those boxes are enabled, the user can drag pins from one box to the other to switch sides and drag pins within a box to switch the pin order.

Bus Pin Editor

The **Bus Pin Editor** tab allows you to create, edit, and delete bus pins from regular pins. The pin names are the terminal names associated with the component model and cannot be changed, but the bus pin names can be anything the user wants.

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 new bus pin**
  - Select the pins needed for the bus pin and click **Combine in to Bus Pin**. Both regular pins and bus pins can be combined to create the new bus pin.
  - Pins can be selected with Ctrl + Click or Shift + click.
  - Pins can be dragged and dropped onto an existing bus pin.
- **To remove pins from a bus pin**
  - Select the whole bus pin or individual pins inside the bus pin and click **Uncombine from Bus Pin**.
  - Pins can be dragged out of the bus pin to the root level to remove on the bus pin.
- **Re-ordering Pins**
  - The pins inside a bus pin can be reordered by dragging them to the appropriate location. The order is important since the first pin with index 0 connects to bus[0] when a bus is connected, and so on.
  - When either the root node (red symbol) is selected OR a bus pin is selected, Sort features become available. Click the buttons to order the pins and bus-pins immediately under the selected node by name in either alphanumeric or reverse-alphanumeric order.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

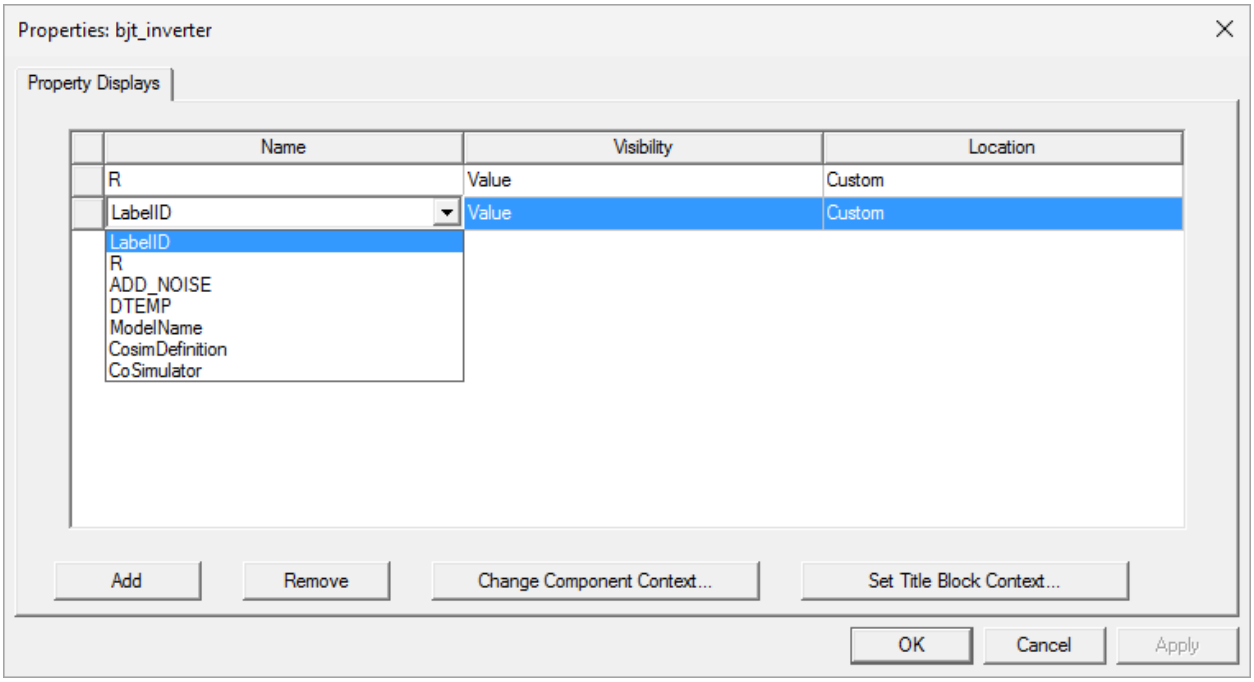
- **Searching the Tree**
  - The tree can be filtered by the pin name by typing in the filter box.
  - The filter text supports wildcard search (*e.g.*, "P*1" should search for everything that begins with a "P" and ends with a "1").
  - Search works for both the root level and leaf level of pins (e.g., if a search matches the pin name under a bus pin, the bus pin is displayed in the search result).

Committing Changes

Clicking **OK** in the window commits changes to the component symbol made in all three tabs.

Symbol Property Display Setup

The **Symbol** > **Property Display Setup** command opens a **Properties** window in which you can add or modify how the names, values, and locations of properties associated with the component or object (such as a title block) are represented by the current symbol.



The Symbol Property Displays Window

This window lists symbol display properties and controls their presence, visibility, and position. Every entry in this window corresponds to a property on a component associated with the symbol in question. Whoever creates or edits a component determines which property to add. Some properties such as [LabelID] can appear in multiple components. The component is set

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatically if you are editing an existing symbol. To change the reference component, click **Change Component Context**.

> **Note:** [LabelID] may display as **[InstanceName]** or *** in the Symbol editor.

Columns in the grid are:

- **Name** – Click in this cell to type a property display name, or to select an existing property display from a list. The component selected via **Change Component Context** determines which property displays are listed for selection.
- **Visibility** – Click in this cell to set the visibility (**None**, **Name**, **Value**, or **Both**) of a property display.
- **Location** – Click in this cell to set the location (**Left**, **Top**, **Right**, **Bottom**, **Center**) of the property display relative to the symbol center.

> **Note:** You can further position a property display by rotating it (**Ctrl**+**R**), or by dragging it to another location. When you do this for a given property display, its Location value is set to **Custom**.

The actions available in this window are:

- Click **Add** to add a new property display entry to the list.
- Click **Remove** to remove the selected property display.
- Click **Change Component Context** to open the **Select Definition** window, to select the component definition on which property displays are based. The properties of the selected component are then available in the **Name** cell properties list.

> **Note:** Because a symbol can be used by multiple components with differing properties, any property name can be entered by the user. If a component using the symbol does not have a property with that name, the symbol property isn't used in that component.

- Click **Set Title Block Context** to make the list of default page properties, and others as specified, available for display. Default properties include: ProjectPath, Project, Design, Title, Author and Date. This context is selected by default if the symbol has three or more propdisplays for default page properties.

Symbol Grid Setup window

The **Symbol** > **Grid Setup** window sets visibility, colors, resolution, and snapping for the Symbol Editor's alignment grid. It contains the following controls.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Major** – Specifies the spacing of the major grid lines (default in mils). To change the color of the major grid lines, click the **Major** color button, specify a color in the **Color** window, then click **OK**.
- **Minor** – Specifies the spacing of the minor grid lines (default in mils). To change the color of the minor grid lines, click the **Minor** color button, specify a color in the **Color** window, then click **OK**.
- **Show Grid** – Toggles grid line visibility.
- **Snap to Grid** – Controls whether graphics and text placed on the grid automatically snap to the nearest grid intersection.
- **Background Color** – To set the background color used for symbol editing, click the color box to open its window, specify a color, then click **OK**.
- **Save as Default** – Save the current **Grid Setup** values as defaults for future Electronics Desktop sessions.
- **Defaults** – Restore all **Grid Setup** settings to their defaults.
- **OK** – Commit changes and close the window.
- **Cancel** – Close the window without committing changes.

Note:

- To ensure electrical connectivity among schematic elements, placed symbol pins snap to a 100-mil grid (2.54-millimeter) regardless of the graphical Major and Minor grid line settings. This snapping cannot be deactivated, and the spacing of the connectivity grid cannot be adjusted.
- The units used for grid line spacings are controlled by the **Tools** > **Options** > **General Options** window.
- Use of the **Minor** grid setting varies depending upon the active editor. In the Layout Editor, the minor grid-line setting specifies the number of units between each minor grid division. However in the Schematic and Symbol Editors, the **Minor** grid-line setting specifies the number of minor grid lines that appear between major grid lines.

Using the Footprint Editor

The Desktop **Footprint Editor** supports the viewing, modification, and saving of footprint information for components and layout primitives. After modifying a footprint, the information can be saved, used to define a design, or exported.

A footprint is a set of information that defines the planar space consumption, physical orientation, metal usage, and electrical connectivity of a component. Footprint definitions are stored in library files with an extension of **.aflb**. You can access material definitions using the **Footprint Editor**.

Creating or editing a footprint involves using the footprint editor to:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Draw graphical primitives, such as rectangles, circles, and arcs, using options on the Draw menu
- Add pins using the Pin option on the Draw menu
- Add text labels using the Text option on the Draw menu
- Update the current project with the new or revised footprint definition using the **Update Project** option on the **Footprint** menu

You can export a footprint to a footprint library (**.aflb**) file for use in other projects. For information on how to do this, see the Edit Libraries window topic.

Opening the Footprint Editor

You can create a new footprint definition or edit an existing footprint.

Creating a New Footprint

To create a new footprint, do one of the following:

From the Tools Menu:

1. Open the **Tools** > **Edit Libraries** > **Footprints** window, then click **Add Footprint**.

   The **Definition Name** window opens and allows you to specify the name for the new footprint.

2. Type a name for the footprint into the **Enter the name for this new Footprint** group box.
3. Click **OK** or press **Enter**.

   The **Choose Technology** window opens and allows you to specify the original layers of the definition

4. Double-click the entry for the new footprint, or click **Edit Footprint**.

From the Project Manager:

1. Select **Add Definition** on the right-click menu in the **Footprints** folder of the **Project Manager** window.

   The **Definition Name** window opens and allows you to specify the name for the new footprint.

2. Type a name for the footprint into the **Enter the name for this new Footprint** group box.
3. Click **OK** or press **Enter**. The **Choose Technology** window opens and allows you to specify the original layers of the definition

Clone an existing footprint:

1. Open the **Tools** > **Edit Libraries** > **Footprints** window.
2. Locate and select an existing footprint on which to base your new footprint.

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 **Clone Footprint(s)** to open the **Get Name** window.
4. Type a name for the new footprint into the **Enter the name for this new Footprint** group box.
5. Click **OK** or press **Enter**.
6. Double-click the entry for the new footprint, or click **Edit Footprint** to open the **Footprint Editor**.

Editing an Existing Footprint

To edit an existing footprint, do one of the following:

- From the **Project Manager** window, expand the **Definitions** /**Footprints** subfolder for the project that contains the footprint you want to edit. Double-click the entry for the footprint you want to edit, or right-click the entry and select **Edit Footprint**.
- Open the **Tools > Edit Libraries > Footprints** window, then locate and select the Project version of the footprint that you want to edit. Click **Edit Footprint**, or double-click the selected entry.
- Select the footprint you wish to edit and click **Edit Footprint** in the shortcut menu.

The footprint editor runs and opens the selected footprint for editing.

Load a Footprint into the Current Project

To load a footprint into the current project:

1. With a new local project open, select **Tools** > **Edit Libraries** > **Footprints**.
   This opens the Edit Libraries window.
2. Select a component in the Name column, for instance, BENDA, and click **Edit Footprint**.
   This loads the component into the local project.

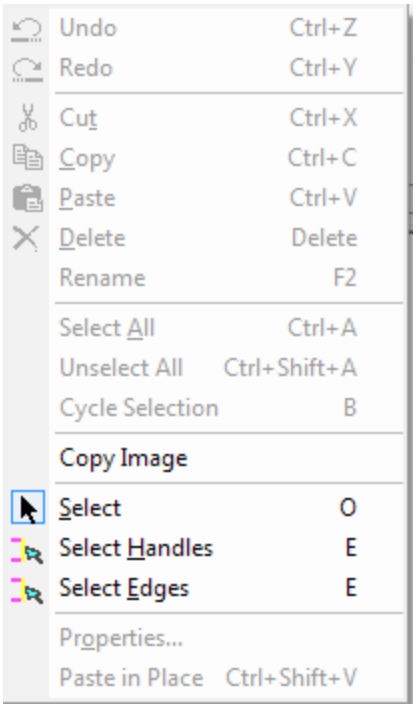
Footprint Editor Operations

The footprint editor opens when a footprint is created or modified. Like the schematic and symbol editors, the footprint editor is a graphical tool that keeps the placement of footprint elements uniform by snapping them to points on a grid.

With the footprint editor, add and manipulate graphical primitives, text labels, pins, vias, and padstacks, and/or adjust the resolution, color, and visibility of the editor grid. Once changes are made, update the current project with any changes to the footprint. Options related to these operations are available on the **Footprint** menu and the Footprint Draw menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Footprint Edit Menu



The fields on this menu become active as appropriate to the type of footprint being edited.

Undo <*last edit operation*> – Undo the last edit operation.

Redo <*last edit operation*> – Redo the last edit operation.

Cut – Delete the selected object, and retain a copy for pasting into a layout in the same application.

Copy – Create a local copy for pasting into a layout in the same application.

Paste – Put the object on the last cut or copy into the layout.

Delete – Delete the selected object without retaining a copy.

Rename – Click to rename the selected object in the layout.

Select All – Select all objects in the layout.

Unselect All – Unselect all objects in the layout.

Cycle Selection – Cycle selection of two or more objects that overlap. Each time you click **Cycle Selection**, a different one of the overlapping objects 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.

Copy Image – Create a global copy of the selected objects on the clipboard for pasting into a different application.

Select – Sets cursor mode to selection of objects.

Select Handles – Sets cursor mode to selection of object handles.

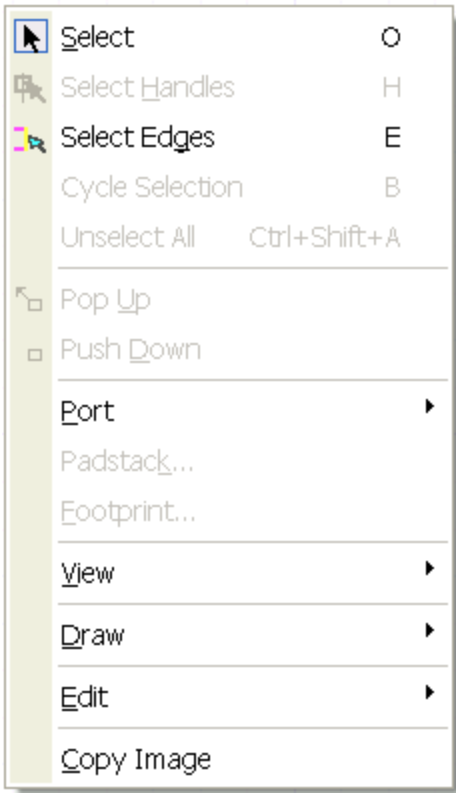
Select Edges – Sets cursor mode to selection of object edges.

Properties – Opens the **Properties** window to view and edit the properties of the selected object.

Paste in Place – Paste object to same location, without any X,Y offset/displacement.

Footprint Shortcut Menu

When a footprint design is active, right-click to open the shortcut menu for the Footprint Editor, similar to the following:

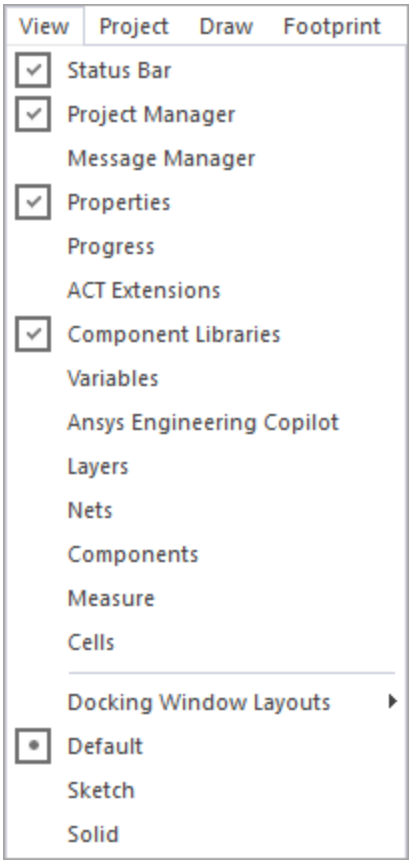


The fields on this menu become active as appropriate to the type of footprint being edited.

Footprint View Menu

When the Footprints Editor is active, the Toolbar View menu has 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.



The first portion of the **View** menu controls the display of the windows and panes available for the Footprint Editor. Click on an entry to put a check mark beside it, which indicates that it is visible. The following options are available:

- **Status Bar**
- **Project Manager**
- **Message Manager**
- **Properties** window
- **Progress** window
- **ACT Extensions**
- **Component Libraries**
- **Variables** – project and design variables
- **Ansys Engineering Copilot**
- **Layers window**
- **Nets** window
- **Components** window
- **Measure** – Open a window for measuring the position, length, and area of elements

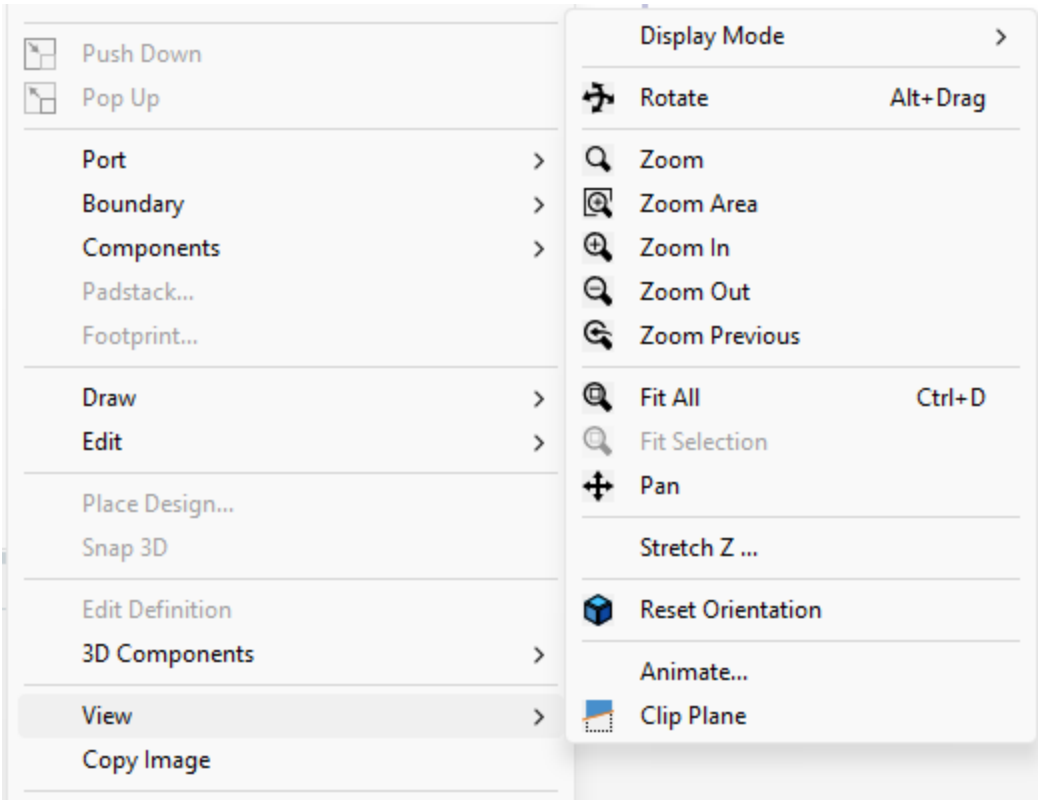
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Cells** – Opens the **Cells** window. If the footprint has hierarchy, the dimensions will be visible in the **Cells** window.

The second portion of the View menu controls the rendering of the footprint:

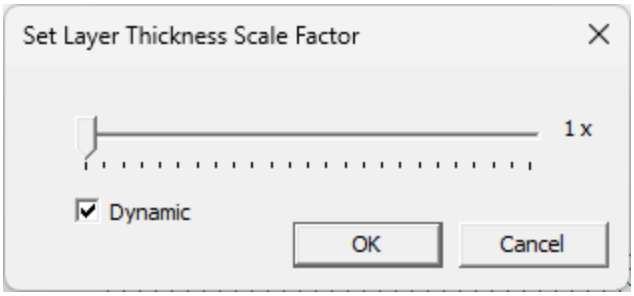
- **Default** – Rendering mode that honors transparency settings configured in stackup
- **Sketch** – Rendering mode very similar to wireframe
- **Solid** – Rendering mode that honors the wireframe settings per layer

If you access the View menu by right clicking on a footprint in the Modeler window, you will see the following menus:



In addition to the standard **Rotate**, **Zoom**, **Fit**, and **Pan** options, this menu includes a **Stretch Z** option, which opens the Set Layer Thickness Scale 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.



Dragging the slider applies a Z scaling factor to the current view. If **Dynamic** is selected, the scaling happens in real time; otherwise, it is applied when you click **OK**.

Footprint Draw Menu

The footprint editor **Draw** menu presents options for drawing graphical primitives and text, and manipulating selected objects. The **Draw** menu options include:

Primitive > Arc

This option, also available on the toolbar, initiates creation of an arc. To draw an arc, click the footprint editor grid at the two points that determines the arc's ends, and then drag the arc to the appropriate radius.

Once you have created an arc, adjust its radius or move its endpoints by clicking the arc to select it, and dragging the appropriate handle.

Primitive > Circle

This option, also available on the **Footprint Draw** toolbar, initiates creation of a circle. To draw a circle, click the footprint editor grid to select a center point, then drag the circle to the appropriate diameter.

- Click the circle to select it, then drag one of its handles.
- Double-click the circle to open its **Properties** window, type a new value for the **Radius** parameter, and click **OK**.

Primitive > Line

This option, also available on the **Footprint Draw** toolbar, initiates creation of a polyline with one or more segments. To draw a line:

To complete the line, do one of the following after defining its final segment:

- Press the **Spacebar**.
- Right-click and click **Finish**.

After you have completed a line, change the endpoints of its segments 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.

- Select the line and drag the appropriate handle(s).
- Double-click the line to open its **Properties** window, type new values for the parameters you want to change, and click **OK**.

Primitive > Rectangle

This option, also available on the **Footprint Draw** toolbar, initiates creation of a rectangle. To create a rectangle, click the editor grid to specify the position of one corner, then drag the rectangle to the appropriate size.

Once you have drawn a rectangle, Edit its height and width, the position of its center, and its angle (its rotation, in degrees, relative the handles of its bounding box) as follows:

- Double-click the rectangle, then edit its properties in the **Properties** window.

Primitive > Polygon

This option, also available on the **Footprint Draw** toolbar, initiates creation of a polygon. To create a polygon:

1. Click the footprint editor grid to specify the position of one vertex, then click wherever you want to place additional vertices.
2. To complete a polygon, specify the position of its final vertex, then do either of the following:

Once you have drawn a rectangle, Edit its vertex positions and other properties, including its fill style, as follows:

- Click the rectangle, then edit its properties in the **Properties** window.
- Double-click the rectangle, then edit its properties in the **Properties** window.

Primitive > Text

This option, also available on the **Footprint Draw** toolbar, adds to the footprint an editable “Default text” label in 12-point Arial. To edit the default text string immediately after placement, type the text you want, then press **Enter** or click elsewhere in the editor grid.

To change just the text of an existing label:

1. Click the label.
2. Click the label again to open its text for editing.
3. Press **Enter** or click elsewhere in the editor grid.

To change other properties of a label, including its font and size:

1. Click the label and view its properties in the **Properties** window, or double-click the label and view its properties in the **Properties** window.
2. Click in the **Value** cell for the property you want to modify.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Modify the value.
4. Click **OK**, or click in another **Value** cell to commit the change and keep editing values.

Primitive > 3D Line

Draw an object with one or more straight segments. For more information, see Drawing a 3D Line Segment.

Void > Circle

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Circle command. One object can have multiple voids, but each void can only be associated with one “parent” object.

Void > Line

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Line command. One object can have multiple voids, but each void can only be associated with one “parent” object.

Void > Rectangle

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Rectangle command. One object can have multiple voids, but each void can only be associated with one “parent” object.

Void > Polygon

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Polygon command. One object can have multiple voids, but each void can only be associated with one “parent” object.

Port > Create

Click the Port > Create command to create a port.

Port > Remove

Click the Port > Remove command to remove a port.

Pin

This option, also available on the **Footprint Draw** toolbar, initiates placement of a footprint pin with a default stem length of 10. You can rotate a pin once you’ve started to place it by iteratively pressing R until the pin is oriented to your liking. Click in the editor grid to finish placing the pin.

Via

This option, also available on the **Footprint Draw** toolbar, initiates placement of a through hole (via) of the default size (1mm).

Handle

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option, also available on the **Footprint Draw** toolbar, allows you to create an explicit footprint handle that can later be used to resize or reshape the footprint based on the handle's reshaping rules.

3D Structure

Opens a menu that allows you to create and perform operations on a 3D structure in the Drawing Region.

Coordinate System

Opens a menu that allows you to create and perform operations on a coordinate system in the Drawing Region manage. For more information see Coordinate Systems in the **Layout Editor** topic.

Rotate

This option, also available on the **Footprint Draw** menu or by pressing **Ctrl**+**R** on the keyboard, rotates a selected object or group of objects 90° to the left.

Flip Vertical

This option, also available on the **Footprint Draw** menu, flips a selected object about the X axis.

Flip Horizontal

This option, also available on the **Footprint Draw** menu, flips a selected object about the Y axis.

Reverse Line

Reverse selected objects in the Drawing Region.

Align...

Position Relative

When a relative coordinate system (CS) and an object independent of that CS are selected, the "Draw > Position Relative" command positions the CS relative to that object.

Clear Position Relative

When a relative coordinate system (CS) and an object are selected, the "Draw > Clear Position Relative" command positions the CS unrelative to that object.

Duplicate

Duplicate a selected object in the Drawing Region.

Expand

Expand selected objects in the Drawing Region.

Split Polygon Region

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Split selected polygon regions in the Drawing Region.

Geometry Healing

Geometry heal selected objects in the Drawing Region.

Stitch Lines

Stitch together connected/crossing lines into polygons and lines.

Defining Footprint Handles

Handles appear around the perimeter of a selected element as small squares in the selection color. If the cursor is moved over one of these handles, it changes to indicate its proximity to the handle. When clicking on a handle, the user can then resize or reshape the element based on the handle's reshaping rules.

Component footprints can also have handles which can be specified in the footprint editor. These handles can likewise be used to reshape a component footprint, and at the same time altering one or more of its electrical parameters. In other words, the user can use handles to directly manipulate an element's geometry corresponding to electrical parameters.

Defining a Handle

Open the footprint editor for the footprint appropriate. From the **Draw** menu, choose **Handle**. Next, click to place the handle anywhere in the drawing, exact placement does not matter.

Editing Handle Location

Next, open the property window (**Edit > Properties**). There is one property listed: location. For this property, edit the X & Y locations to place the handle in the correct position based on the parameters.

Specifying Parameters to Change

Next, click **Add** in the **Property** window. In the responding window, specify the name of the electrical parameter, and the expression, based on the X & Y location of the handle that sets this parameter. For example:

Edit the footprint for the MSTRL component. Place a handle that is on the top edge of the line. The component has the parameters P for length and W for width. The handle has its location (X,Y) set to: (P/2, W/2). One property is added called "W" and its expression is "abs(Y*2)". The "abs" (absolute value) of "Y*2" ensures that as Y approaches zero, the line width stays positive. The two entries in the window should look like the following:

Location: P/2, W/2

W: abs(Y*2)

How it works

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 user drags the handle, for each new location of the cursor, each of the handles properties are recalculated for that new position. For the TRL example, the y position of the top-side handle is used to set the line's "w", which of course is twice that of y. After the new parameter values are calculated, the handle's own position is recalculated based on the updated parameters and the expressions for its location. Thus, as the cursor goes up, the line gets wider, and the handle tracks with the new width.

Using Scripts to Define Footprints

Footprints in Layout can use scripts and primitives in defining the geometry needed. Scripts can also move and/or create ports. Scripts can be written in any Windows supported languages.

When Layout needs to retrieve footprint geometry, it invokes the script, passing an object called LayoutHost. This object provides the interface to the needed functions within Layout necessary for creating geometry and accessing information within Layout. The script then calls the member functions of this host object to define the footprint appropriate.

Documentation on scripting within Windows can be found at:
http://msdn.microsoft.com/scripting/.

Layout Host Object

LayoutHost is an object within Layout which provides access to necessary Layout functions for use in a script. This topic describes the interface for this function.

LayoutHost Properties

Pars

Returns an ElementPars object. This property is used to set/get the component parameters declared in the footprint.

Example:

var pars = LayoutHost.Pars;

LayoutPars

Returns an ElementPars object. This property is used to set/get the locally defined parameters in the footprint.

Example:

var layPars = LayoutHost.LayoutPars;

LengthUnits

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Returns a string. This property is used to set/get the internal length units in the current script. By default, length units in the script are meters. When this property is changed, any subsequent calls to the LayoutHost are interpreted in the new units.

Example:

LayoutHost.LengthUnits(“mm”);

MID

A constant used to query an edge for its mid point.

Example:

var x = rect.Edge(1).X(LayoutHost.MID);

START

A constant used to query an edge for its start point.

Example:

var x = rect.Edge(1).X(LayoutHost.START);

END

A constant used to query an edge for its end point.

Example:

var x = rect.Edge(1).X(LayoutHost.END);

LayoutHost Methods

The following LayoutHost methods are available for use in **Electronics Desktop**.

LayoutHost General Methods

GetDefinitionType()

Returns an integer which indicates the type of shape used to define a shape-based footprint. This type of footprint allows the user to draw a shape such as a rectangle or polygon , the script uses this definition to create a footprint. The method GetDefinitionPoints is then used to retrieve the points input by the user to define the shape’s outline.

Example:

var shapeType = LayoutHost.GetDefinitionType();

Possible shapes are:

0: Single point

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1: Rect

2: Polyline

3: Polygon

GetDefinitionPoints()

Returns a PointsObject containing the points defining an outline for a shaped-based footprint. This method is used along with GetDefinitionType to retrieve the information needed regarding the base shape used in order to define the footprint.

Example:

```
var outline = LayoutHost.GetDefinitionPoints();
```

GetLastError()

Returns an integer id regarding the last function call which changes the footprint. Functions which change the footprint include those creating geometry, and those altering pins. A non-zero result indicates an error occurred. Zero is returned for success.

Example:

```
var error = LayoutHost.GetLastError();
```

GetLayerID(name)

Returns an integer id for the given name. If a layer with “name” is not found, then a -1 is returned. The name is not case-sensitive.

Example:

```
var layerID = LayoutHost.GetLayerID(“Top”);
```

CreatePointsObject()

Creates and returns an empty PointsObject. Points can then be added to this object which in turn is passed as a parameter to certain primitive creation functions.

Example:

```
var pts = LayoutHost.CreatePointsObject();
```

LayoutHost Shape Creation Methods

These methods create virtual shapes in the footprint. While these shapes mirror actual primitives, they are drawn only and are not selectable or editable. All length values are in the current units of the footprint.

NewArc(layer, cx, cy, r, ang1, ang2, lw)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creates an arc in the footprint, returns a geometry object.

Parameters:

Layer: In integer id retrieved from GetLayerID

Cx: Center x coordinate

Cy: Center y coordinate

R: Radius of arc centerline

Ang1: Start angle of arc (in degrees)

Ang2: Stop angle of arc going CCW

Lw: Line width of arc

Example:

var geom = LayoutHost.NewArc(layerID, 0, 0, r, 0, 90, w);

NewArcLine(layer, cx, cy, r, ang1, ang2, lw, end)

Creates an arc line in the footprint, returns a geometry object.

Parameters:

Layer: In integer id retrieved from GetLayerID

Cx: Center x coordinate

Cy: Center y coordinate

R: Radius of arc centerline

Ang1: Start angle of arc (in radians)

Ang2: Stop angle of arc going CCW

Lw: Line width of arc

End: End style. Possible end styles are: flat, extended, round.

Example:

var geom = LayoutHost.NewArcLine(layerID, 0, 0, r, 0, 90, w, “flat”);

NewCircle(layer, cx, cy, r)

Creates a circle in the footprint, returns a geometry object.

Parameters:

Layer: In integer id retrieved from GetLayerID

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cx: Center x coordinate

Cy: Center y coordinate

R: Radius of circle

Example:

var geom = LayoutHost.NewCircle(layerID, 0, 0, r);

NewLine(layer, points, lw, join, end)

Creates a polyline in the footprint, returns a geometry object.

Parameters:

Layer: In integer id retrieved from GetLayerID

Points: A PointsObject populated with the points appropriate

Lw: Linewidth

Join: Join style. Possible join styles are: corner, miter, round.

End: End style. Possible end styles are: flat, extended, round.

(Passing an empty string defaults to round for Join or End style.)

Example:

var geom = LayoutHost.NewLine(layerID, pts, w, “corner”, “flat”);

NewPoly(layer, points)

Creates a polygon in the footprint, returns a geometry object.

Parameters:

Layer: In integer id retrieved from **GetLayerID**.

Points: A PointsObject populated with the points appropriate

Example:

var geom = LayoutHost.NewPoly(layerID, pts);

NewRect(layer, cx, cy, width, height, angle)

Creates a rectangle in the footprint, returns a geometry object.

Parameters:

Layer: In integer id retrieved from GetLayerID

Cx: Center x coordinate

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cy: Center y coordinate

Width: Width of rect

Height: Height of rect

Angle: Rotation angle to draw rect (in radians)

Example:

var geom = LayoutHost.NewRect(layerID, 0, 0, w, h, 0);

NewText(layer, text, x, y, font, size, angle, just)

Creates text in the footprint, returns a geometry object.

Parameters:

Layer: An integer id retrieved from GetLayerID

Text: The text string

X: Text x placement position

Y: Text y placement position

Font: Text font; may be left blank. On system layers, a system font must be used. Otherwise, specify a plotter font (e.g., "RomanDuplex").

Size: Text height (in length units for a plotter font; in points for a system font)

Angle: The text angle (in radians)

Just: Text justification relative to the placement position: "LeftTop", "LeftBase", "LeftBottom", "CenterTop", "CenterBase", "CenterBottom", "RightTop", "RightBase", "RightBottom"

Example:

var text = LayoutHost.NewText(layer, "Text", 2, -1, "", 1, 0, "LeftBase");

LayoutHost Relative Shape Placement Methods

These methods place shapes relative to each other

SnapEdges(edge, edge_pos, to_edge, to_edge_pos);

Places one geometry object against another so the specified edges are parallel and points touching

Parameters:

Edge: Specifies both the object to be moved and the edge to snap.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edge_pos: The position on the source edge (the object being moved) that is to be snapped to the target edge; specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

To_edge: Specifies the target object and edge (this object is not moved)

To_edge_pos: The target snap point. The source edge and position is snapped to this location; specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

Example:

// Snap 'rect' so as that edge 1 is placed against edge 2 of 'rect2'

LayoutHost.SnapEdges(rect.Edge(1), LayoutHost.MID, rect2.Edge(2), LayoutHost.MID);

SnapEdgesWithOffset(edge, edge_pos, to_edge, to_edge_pos, x_off, y_off, rel_angle);

Same as for SnapEdges but the target snap point is specified by an edge position with an offset and angle. Associated with the target edge position is a normal vector and a tangent vector. The normal vector is perpendicular to the edge and points outwards (to the right in polylines). The tangent vector points in the direction of the edge. These two vectors define the local coordinate system in which x_off, y_off, and angle are specified.

Parameters:

Edge: Specifies both the object to be moved and the edge to snap.

Edge_pos: The position on the source edge (the object being moved) that is to be snapped to the target edge; specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

To_edge: Specifies the target object and edge (this object is not moved)

To_edge_pos: The target snap point. The source edge and position is snapped to this location; specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

X_off: An offset along the normal vector for the target edge.

Y_off: An offset along the tangent vector for the target edge.

Rel_angle: An additional rotation (specified as a counter-clockwise rotation in radians) relative to the normal vector for the target edge.

Example:

// Snap 'rect' so as that edge 1 is offset 1 unit from edge 2 of 'rect2'

LayoutHost.SnapEdgesWithOffset(rect.Edge(1), LayoutHost.MID, rect2.Edge(2), LayoutHost.MID, 1, 0, 0);

LayoutHost Port and Pin Methods

These methods create and move ports/pins as needed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MovePort(name, cx, cy, angle)

Moves an existing port of that name. All scripted footprints must call MovePort for every pin in the footprint. This call should be made prior to defining any port geometry.

Parameters:

Name: Name of the port

Cx: Center x coordinate

Cy: Center y coordinate

Angle: Rotation angle to draw port (in radians)

Example:

LayoutHost.MovePort("1", x, y, 2*Math.tan(theta));

SnapPort(name, edge, edge_pos, rel_angle)

Moves a port to the specified edge position. The angle is relative to the normal vector (perpendicular to the edge and pointing outwards, or to the right in poly-lines).

Parameters:

Name: Name of an existing port

Edge: The edge on which the port is to be placed

Edge_pos: The position on the edge where the port is to be placed; specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

Rel_angle: An additional rotation (specified as a counter-clockwise rotation in radians) relative to the normal vector for the edge.

Example:

LayoutHost.SnapPort("1", rect1.Edge(1), LayoutHost.MID, 0);

SetPortLayers(name, start_layer, end_layer)

Sets the layer range (and hence placement) for a port

Parameters:

Name: Name of the port

Start_layer: An integer id retrieved from GetLayerID

End_layer: An integer id retrieved from GetLayerID

Example:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

var top = LayoutHost.GetLayerId(“top”);

var bottom = LayoutHost.GetLayerId(“bottom”);

LayoutHost.SetPortLayers(“a”, top, bottom);

AddPortEdge (layer, name, points)

Defines the edge of the port to the polyline defined by “points”. Defining the port edge is necessary for co-simulation of a scripted footprint. Note, if you define the port edge, you must also have set the port position via “MovePort” first. Multiple edges may be added by calling this function repeatedly.

The points for the edge should be defined with the port as the origin. For example, if the edge is a vertical line 10 mils long running thru the port, the two points defining the edge is (0, 5) & (0, -5), regardless of the location of the port itself.

Parameters:

Layer: In integer id retrieved from GetLayerID

Name: Name of the port

Points: A PointsObject populated with the points appropriate

Example:

LayoutHost.AddPortEdge(layerID, “n1”, pts));

SetPortWidth (layer, name, w)

Defines the edge of the port to be a single line segment of width “w”, centered on the port position, and rotated to the port angle. This function is an alternate means of defining a port edge. Defining the port edge is necessary for co-simulation of a scripted footprint. Note, if you define the port edge, you must also have set the port position via “MovePort” first.

Parameters:

Layer: An integer id retrieved from GetLayerID

Name: Name of the port

W: Width of the port edge

Example:

LayoutHost.SetPortWidth(layerID, “n1”, w));

LayoutHost Via and Padstack Methods

NewVia(name, start_layer, end_layer, cx, cy, padstack, diameter)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creates a via in the footprint and returns it as a via object.

Parameters:

Name: Name of the via

Start_layer: Integer id retrieved from GetLayerID

End_layer: Integer id retrieved from GetLayerID

Cx: Center x coordinate

Cy: Center y coordinate

Padstack: Name of the padstack to use. Passing an empty string (“”) results in “No Pad SMT East” by default.

Diameter: Diameter of the via

Example:

var via = LayoutHost.NewVia(“via_name”, layerID1, layerID2, x, y, “Planar EMVia”, diameter);

SnapVia(via, to_edge, to_edge_pos, x_off, y_off, rel_angle);

Moves a via to the specified edge position. The target snap point is specified by an edge position with an offset and angle. Associated with the target edge position is a normal (perpendicular to the edge and pointing outwards, or to the right in poly-lines) and a tangent vector (pointing in the direction of the edge). These two vectors define the local coordinate system in which x_off, y_off, and angle are specified.

Parameters:

Via: Specifies the via object to be moved

To_edge: Specifies the target object and edge

To_edge_pos: The target snap point. The via position is snapped to this location, specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

X_off: An offset along the normal vector for the target edge.

Y_off: An offset along the tangent vector for the target edge.

Rel_angle: An additional rotation (specified as a counter-clockwise rotation in radians) relative to the normal vector for the target edge.

Example:

// Place ‘via’ 1 unit inside edge 2 of ‘rect’

LayoutHost.SnapVia(via, rect.Edge(2), LayoutHost.MID, -1, 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.

ElementPars Object

This object permits retrieval of parameters & local variables defined in the footprint for use in a script. Retrieval of this object was discussed under LayoutHost.

Count

Contains the number of parameters in this object. This property is read-only.

Example:

```
var pars = LayoutHost.Pars;

var numPars = pars.Count;
```

Item

Contains the given parameter in this object. The item may be retrieved by index (zero based) or name. The name is not case sensitive. This property is read-only.

Example:

```
var pars = LayoutHost.Pars;

var w = pars.Item(0);

var p = pars.Item("p");
```

ElementPars Methods

There are no methods for the ElementPars object.

Related Topics

ElementPars Properties

ElementPars Methods

Points Object

This object permits retrieval of parameters & local variables defined in the footprint for use in a script. Retrieval of this object was discussed under LayoutHost.

Points Object Properties

Count

Contains the number of points in this object. This property is read-only.

Example:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
var pts = LayoutHost.CreatePointsObject();
var numPts = pts.Count; // returns 0 in this case
```

X(index)

Contains the X coordinate for a given index (zero based).

Example:

```
var pts = LayoutHost.CreatePointsObject();
x = 10;
pts.Add(x, y);
x2 = pts.X(0) * 2;
pts.X(0) = x2;
```

Y(index)

Contains the Y coordinate for a given index (zero based).

Example:

```
var pts = LayoutHost.CreatePointsObject();
y = 10;
pts.Add(x, y);
y2 = pts.Y(0) * 2;
pts.Y(0) = y2;
```

Points Object Methods

Add(x,y)

Adds a new point to the end of the object's array.

Example:

```
var pts = LayoutHost.CreatePointsObject();
pts.Add(5, 10);
```

Remove(index)

Removes a point on the object's array, given the (zero-based) index.

Example:

```
var pts = LayoutHost.CreatePointsObject();
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pts.Add(5, 10);

pts.Remove(0);

Geom Object

Geometry objects are returned by the shape creation methods (NewRect, NewCircle, etc.) Geometry objects may be manipulated in a variety of ways through the methods exposed by their interface. All coordinates and length related values are passed/returned in the current default script units. Unless changed through LayoutHost.LengthUnits, all values are by default assumed to be in meters (SI units).

Example:

var rect = LayoutHost.NewRect(layer, 0, 0, l, w, 0);

Geom Object Properties

Layer

Get/Set the layer on which a geometry object is placed.

Example:

rect.Layer = LayoutHost.GetLayerID('top');

Edge(index)

Return the edge object for the edge specified. Edge indexes are zero based and count in a counter-clockwise direction around the object. For rectangles, edge zero is the lower edge. Circles are defined by two arc edges, a lower edge (edge zero) and an upper edge (edge 1); the start/end points of these edges are directly to the left/right of center. Lines and Polygon edges are defined by the user when the object is created. A rotated box giving the text extent is used to define the edges for a text object; edge zero is the line beneath the text.

Example:

var len = poly.Edge(5).Length

Count

Returns the number of edges in the geometry object.

Example:

for (var i = 0; i < poly.Count; ++i)

Geom Object Methods

MoveBy(dx, dy)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Translates the geometry object by the specified offset.

Parameters:

Dx: X offset

Dy: Y offset

Example:

rect.MoveBy(10, 20);

MoveTo(x, y)

Moves the geometry object to the specified location; useful for rectangles, circles and arcs (the center point is moved)

Parameters:

X: New X location

Y: New Y location

Example:

rect.MoveTo(10, 20);

Rotate(x, y, angle)

Rotates the geometry object counter-clockwise about the specified location by the angle specified (in radians).

Parameters:

X: X center

Y: Y center

Angle: Rotation angle in radians

Example:

rect.Rotate(0, 0, Math.PI/4);

Scale(x, y, factor)

Scales the geometry object about the specified location by the factor specified.

Parameters:

X: X center

Y: Y center

Factor: Scale factor (2 implies 2 times, 4.5 implies 4 and a half times, etc.)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example:

rect.Scale(0, 0, 2);

MirrorX(x)

Mirrors the X values of the object (Y values are unchanged) about the specified X value (X values falling on this line are unchanged).

Parameters:

X: Center of reflection (X values on this line are unchanged).

Example:

rect.MirrorX(0);

MirrorY(y)

Mirrors the Y values of the object (X values are unchanged) about the specified Y value (Y values falling on this line are unchanged).

Parameters:

Y: Center of reflection (Y values on this line are unchanged).

Example:

rect.MirrorY(0);

Copy()

Create and return a copy of the geometry object.

Example:

var rect_copy = rect.Copy();

AddVoid(void_geom)

Adds a void object to an existing geometry object.

Example:

Var rect = LayoutHost.NewRect(layer, 0, 0, l, w, 0);

Var void_rect = LayoutHost.NewRect(layer, 0, 0, l/2, w/2, 0);

rect.AddVoid(void_rect);

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edge Object

Edge objects are returned by an Edge query on a Geometry object. Note: edges may be arcs or straight segments.

Related Topics

Edge Object Properties

Edge Object Methods

Edge Object Properties

Edge objects are returned by an Edge query on a Geometry object. Note: edges may be arcs or straight segments.

Example:

var edge = geom.Edge(1);

Length

Return the true edge length (if an arc, the curvature is taken into account).

Example:

var len = geom.Edge(1).Length;

Angle(pos)

Return the angle of the tangent (in radians), relative to the standard X axis, of the edge point specified by ‘pos’. Angles are returned using the usual ‘count-clockwise is positive’ convention. Tangents point in the direction of the edge.

Parameters:

Pos: Edge position; usually specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

Example:

var angle = geom.Edge(1).Angle(LayoutHost.START);

Normal(pos)

Return the angle of the normal (in radians), relative to the standard X axis, of the edge point specified by ‘pos’. Angles are returned using the usual ‘count-clockwise is positive’ convention. Normals are outward facing (or to the right in poly-lines) of the edge.

Parameters:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pos: Edge position; usually specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

Example:

var normal = geom.Edge(3).Normal(LayoutHost.END);

X(pos)

Return the X value of the edge point specified by ‘pos’.

Parameters:

Pos: Edge position; usually specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

Example:

var x = geom.Edge(3).X(LayoutHost.END);

Y(pos)

Return the Y value of the edge point specified by ‘pos’.

Parameters:

Pos: Edge position; usually specified as LayoutHost.START, MID, or END (or a value < 0, 0, or > 0).

Example:

var y = geom.Edge(3).Y(LayoutHost.END);

Edge Object Methods

There are no methods for the Edge object.

Via Object

Via objects are returned by the LayoutHost method NewVia. They may be manipulated in a variety of ways through the methods exposed by their interface. All coordinates and length related values are passed/returned in the current default script units. Unless changed through LayoutHost.LengthUnits, all values are by default assumed to be in meters (SI units).

Example:

var via = LayoutHost.NewVia(“via_name”, layerID1, layerID2, x, y, “EM designVia”, diameter);

Via Object Properties

StartLayer

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Get/Set the start layer of the via.

Example:

via.StartLayer = LayoutHost.GetLayerID('top');

EndLayer

Get/Set the end layer for the via.

Example:

var layer = via.EndLayer;

X

Get/Set the x position for the via.

Example:

v1.X = v2.X;

Y

Get/Set the y position for the via.

Example:

v1.Y = v2.Y;

HoleDiameter

Get/Set the hole diameter for the via. A negative diameter turns off the diameter override and the value on the padstack definition is used.

Example:

v1.HoleDiameter = v2.HoleDiameter;

Rotation

Get/Set the via rotation. The rotation is in radians.

Example:

v1.Rotation = Math.PI/4;

Via Object Methods

ClearLayerMapping(fp_layer)

Clears the mapping between a footprint layer and a layer in the padstack definition.

Parameters:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fp_layer: An integer id retrieved from GetLayerID

Example:

```
var v1 = LayoutHost.NewVia("via1", layer, bottom, 0, 0, "Round 1mm/0.5mm", 0.2);
v1.ClearLayerMapping(layer);
```

ClearLayerMappings

Clears the mapping between all footprint and padstack definition layers.

Example:

```
var v1 = LayoutHost.NewVia("via1", layer, bottom, 0, 0, "Round 1mm/0.5mm", 0.2);
v1.ClearLayerMappings();
```

Copy()

Create and return a copy of the via.

Example:

```
var via_copy = via.Copy();
```

DefaultLayerMapping

Reset the mapping between all footprint and padstack definition layers to the default mapping.

Example:

```
var v1 = LayoutHost.NewVia("via1", layer, bottom, 0, 0, "Round 1mm/0.5mm", 0.2);
v1.DefaultLayerMapping();
```

MirrorX(x)

Mirrors the via position about the specified X value (Y value is unchanged; an X value falling on this line is unchanged).

Parameters:

X: Center of reflection (X values on this line are unchanged).

Example:

```
via.MirrorX(0);
```

MirrorY(y)

Mirrors the via position about the specified Y value (X values are unchanged; a Y value falling on this line is unchanged).

Parameters:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y: Center of reflection (Y values on this line are unchanged).

Example:

via.MirrorY(0);

MoveBy(dx, dy)

Translates the via by the specified offset.

Parameters:

Dx: X offset

Dy: Y offset

Example:

via.MoveBy(10, 20);

MoveTo(x, y)

Moves the via to the specified location; useful for rectangles, circles and arcs (the center point is moved)

Parameters:

X: New X location

Y: New Y location

Example:

via.MoveTo(10, 20);

Rotate(x, y, angle)

Rotates the via counter-clockwise about the specified location by the angle specified (in radians).

Parameters:

X: X center

Y: Y center

Angle: Rotation angle in radians

Example:

via.Rotate(0, 0, Math.PI/4);

Scale(x, y, factor)

Scales the via position about the specified location by the factor specified.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters:

X: X center

Y: Y center

Factor: Scale factor (2 implies 2 times, 4.5 implies 4 and a half times, etc.)

Example:

via.Scale(0, 0, 2);

SetLayers(start_layer, end_layer)

Sets the via layer range

Parameters:

Start_layer: An integer id retrieved from GetLayerID

End_layer: An integer id retrieved from GetLayerID

Example:

var top = LayoutHost.GetLayerId(“top”);

var bottom = LayoutHost.GetLayerId(“bottom”);

via.SetLayers(top, bottom);

SetLayerMapping(fp_layer, ps_layer, pad_type);

Creates or adjusts the mapping between a footprint layer and the padstack definition layer.

Parameters:

fp_layer: An integer id retrieved from GetLayerID

ps_layer: The name of the layer in the padstack definition

pad_type: Unused

Example:

var v1 = LayoutHost.NewVia("via1", layer, bottom, 0, 0, "Round 1mm/0.5mm", 0.2);

v1.ClearLayerMappings();

v1.SetLayerMapping(layer, "template", "");

v1.SetLayerMapping(bottom, "template", "");

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 Padstack Editor

A padstack is a set of information that defines the physical structure and electrical connectivity of a pin or via (the region in which a component pin connects to a layout or a connection between layers). A padstack includes information about the layers involved in making the connection, on the size and shape of the layout area to the type and dimensions of the accompanying hole. Padstack definitions are stored in library files with an extension of **.pslb**.

The Electronics Desktop Padstack Editor supports the viewing, modification, and saving of padstack definition information for physical structure and connectivity. After modifying a padstack definition, the information can then be used to define a via or pin. Padstack definitions can be exported to library files.

To open the **Padstack Editor**, load a project and select **Tools** > **Edit Libraries** > **Padstacks**. When the Edit Libraries window opens, use its controls to open the **Padstack Editor**.

Creating a New Padstack

To create a new padstack, do one of the following:

- Select **Tools** > **Edit Libraries** > **Padstacks**, then click **Add Padstack**.

  The **Definition Name** window opens and allows you to specify the name for the new padstack.

  The **Choose Layout Technology** window opens, and you may browse for a technology file to define the original layers of the definition. If no technology file is chosen, the definition is created with initial layers named **Start, Stop, and Default**.

- Select **Tools** > **Edit Libraries** > **Padstacks**, then locate and select an existing padstack definition on which to base a new definition.

  Search for and select a padstack name and then click **Clone Padstack(s)**.

- From the **Project Manager** window, right-click the **Padstack** folder and select **Edit Library** to open the Padstack tab of the **Edit Libraries** window. Click **Add Padstack** or **Clone Padstack(s)** to open the **Edit Padstack Definition** window.
- In the Project Tree, right-click **Padstacks** and select **Add Definition**.

After a padstack is created, edit the padstack.

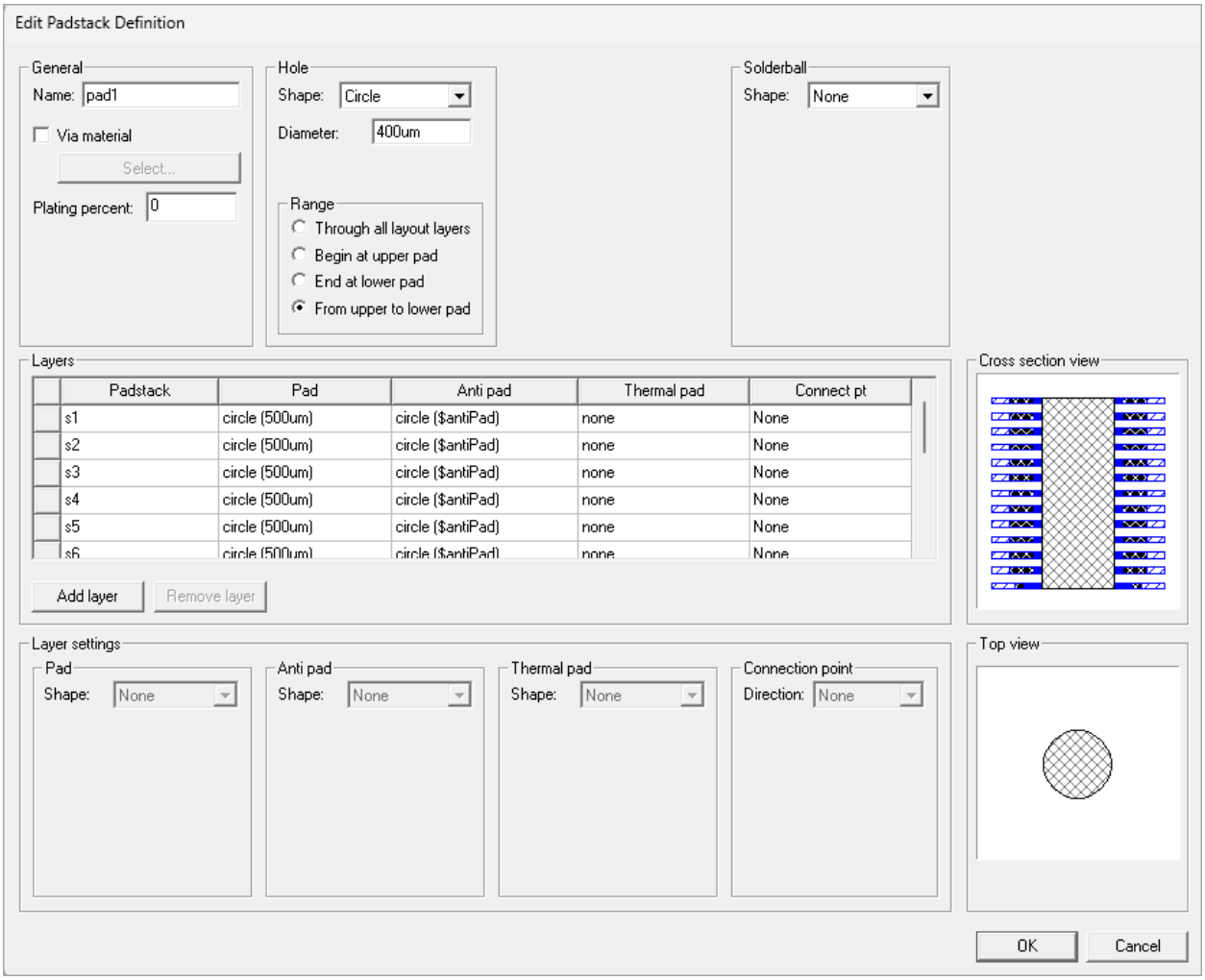
Editing a Padstack

Edit and define a padstack in the **Edit Padstack Definition** window or the **Padstack Usage and Definition** window. Open these windows 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 **Tools** > **Edit Libraries** > **Padstacks**. Find and select the padstack you want to edit, then double-click its name or click **Edit Padstack**.
- In the **Definitions** directory of the **Project Manager** window, right-click the **Padstack** folder and select **Edit Library**. Find and select the padstack you want to edit, then double-click its name or click **Edit Padstack**.
- Select a via or pin that uses the padstack. In the **Properties** window, click **Padstack Usage** to open the **Padstack Usage and Definition** window. When opened from the **Properties** window, information related to the padstack definition is highlighted in blue.

The **Edit Padstack Definition** window is shown below:



Important: Some options may be unavailable or may not appear, depending upon the context and which padstack is being edited.

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.

General

- **Name** – the name of the padstack.
- **Via Material** – the material for plating the via or pin hole. Click **Select** to choose from available materials.
- **Plating Percent** – the percent of the hole's radius that is filled by the plating material. At 100%, plating fills the hole. The **Cross section view** and **Top view** areas show the vertical cross-section and top of the layer stackup. This setting is used by SIwave Solvers and Icepak **Electronics Desktop**.

> **Note:** Zero plating (i.e., 0%) is not recognized and is treated as fully plated.

Hole

- **Shape** – choices are None, Circle, Square, Rectangle and Polygon. The Polygon choice is currently only supported for padstacks; polygon holes may not be defined through the window. Polygons can be modified by selecting a layer and clicking **Edit Polygon** to open the **Polygon Editor**. Selecting a shape enables additional options:
  - **Size / Diameter / XY Size** – specifies the size of the hole.
  - **Range** – can specify the hole range to be:
    - Through all the layout stackup layers
    - Beginning at the upper pad of the padstack and continuing to the lowest elevation of the layout stackup layers
    - Beginning at the highest elevation of the layout stackup layers and ending at the lowest pad of the padstack
    - Beginning at the upper pad and ending at the lowest pad of the padstack

Solderball

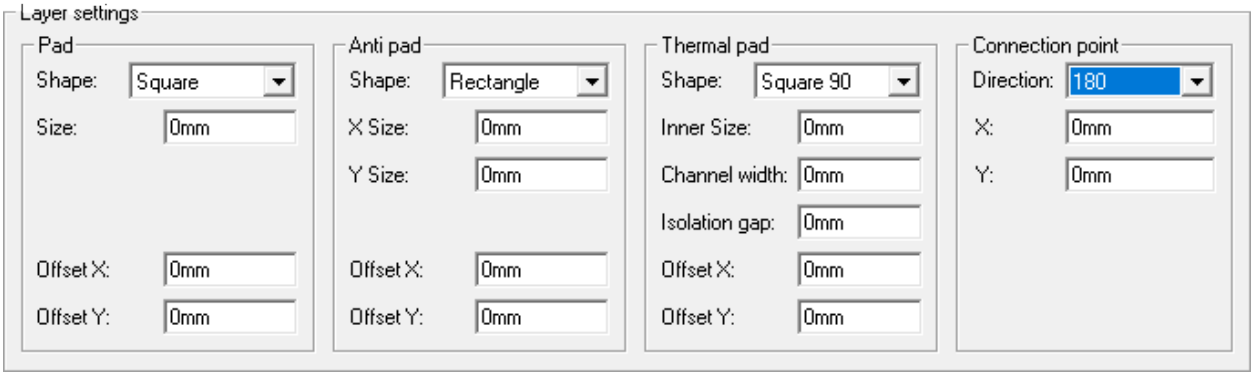
- **Shape** – specifies the shape of the solderball; choices are none, cylinder, and spheroid.
- **Diameter** – specifies the diameter of the solderball.
- **Mid diameter** – for spheroid solderballs, specifies the mid diameter.
- **Solder** – click to open the **Material Editor** for the solderball in order to specify the material used.
- **Connection** – select to specify the location of the solderball either above or below the padstack hole.

Layers

Layer names are used when a padstack definition is used in a via or pin. A default mapping of padstack definition layers to layout layers occurs based on the layer names.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- None of the **Layers** group box grid control cells are directly editable.
- When one or more rows in the **Layers** group box is selected, the **Layer settings** group box is enabled and can be used to change the pad settings for selected rows.



- **Add Layer** inserts a new layer above the selected row.
- **Remove Layer** removes the selected row(s). You may also rearrange rows to be any order.

Layer Settings

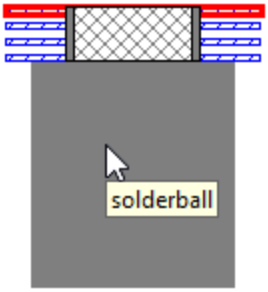
The **Layer settings** area displays pad settings for padstack definition layers. The Polygon choice is currently only supported for imported padstacks; polygon pads, anti pads, and thermal pads may not be defined through the window. Dimension controls appear to support the shape choice for each pad. Controls to specify the pad offset are also available.

- **Pad** and **Antipad** shape choices are None, Circle, Square, Rectangle, Oval, Bullet, N-Sided Polygon, and Polygon. Polygons can be modified by selecting a layer and clicking **Edit Polygon** to open the **Polygon Editor** .
- **Thermalpad** shape choices are None, Polygon, Round45, Round90, Square45, and Square90. Polygons can be modified by selecting a layer and clicking **Edit Polygon** to open the **Polygon Editor**.
- **Connection point** allows you to choose a direction in degrees or specify None. There are also controls to specify the X,Y location of the connection point.

Cross Section View

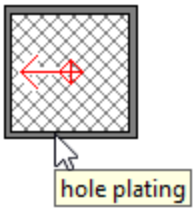
This area displays all definition layers and their pads from a side view. Selected layers are highlighted. Hover the mouse over this area to see tooltips indicating padstack components.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



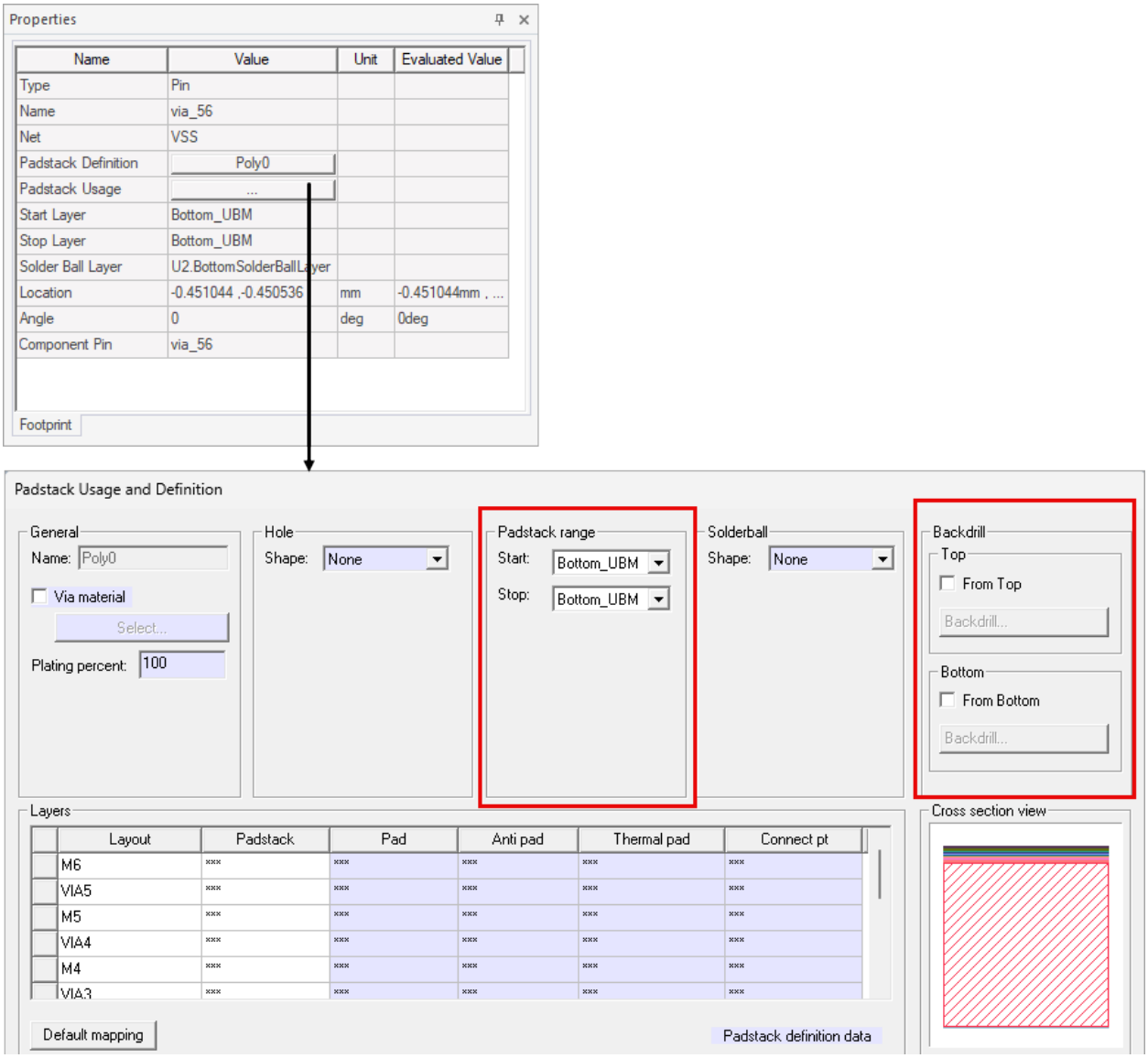
Top View

This area displays the pads of selected layers from a top down perspective. Hover the mouse over this area to see tooltips indicating padstack components.



If the padstack properties are opened from the **Properties** window, the **Padstack Usage and Definition** window opens and includes additional parameters for Padstack Range and Backdrill:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Padstack Range

This area contains controls to specify the **Start** and **Stop** layers for the padstack.

Backdrill

Backdrilling creates empty holes on one or both sides of a via. From the **Backdrill** group box, define backdrills from a selected via's top layer, bottom layer, or both. Refer to Backdrilling. Backdrills are rendered in the **Cross section view** and **Top view** group boxes as empty space. In the following script example, the backdrill is recorded as a block:

```
["NAME:Backdrill Top Diameter", "MustBeInt:=", false, "Value:=",
"0.4mm"]
```

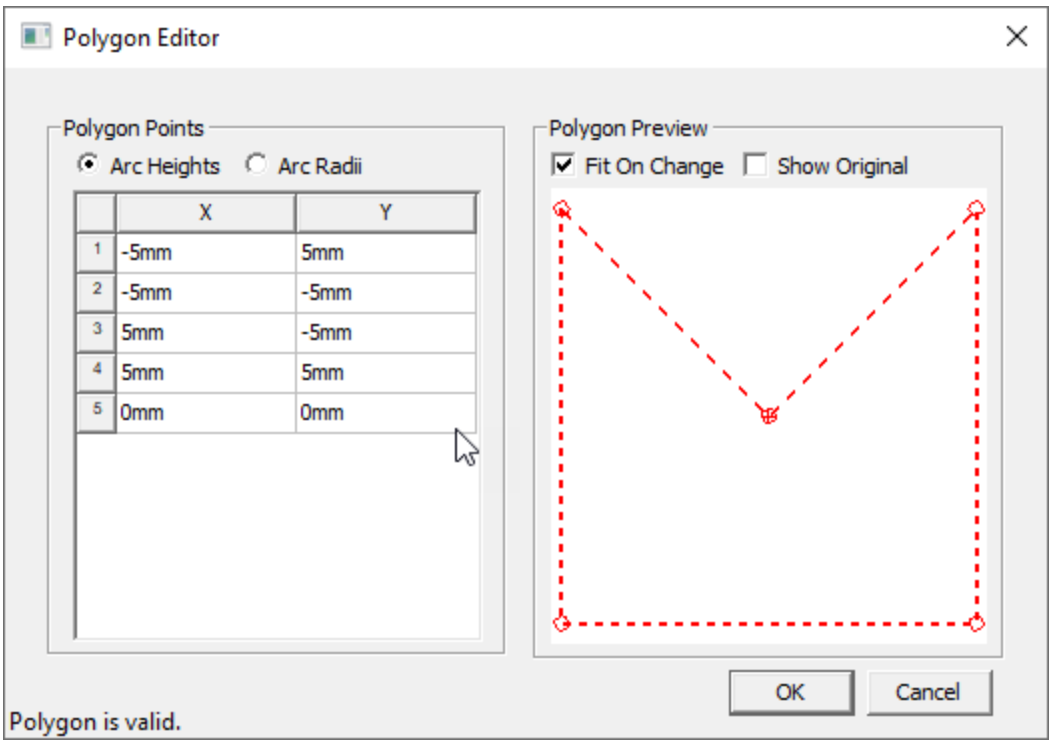

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **From Top** – define a backdrill that starts on the selected via's top layer. Check the adjacent box to activate or remove the check to deactivate. Click the associated button to proceed to the **Backdrill Definition** window.
- **From Bottom** – define a backdrill that starts on the selected via's bottom layer. Check the adjacent box to activate or remove the check to deactivate. Click the associated button to proceed to the **Backdrill Definition** window.

Refer to Define Backdrill Parameters.

Editing a Polygon in a Padstack

Pads, antipads, thermal pads, and holes can have polygon shapes that can be modified in the Polygon Editor, which can be opened by selecting a layer and clicking **Edit Polygon**. This editor lists the points of the polygon in a grid and displays a diagram of the polygon. The **Polygon Editor** is available on the **Edit Padstack Definition** and the **Padstack Usage and Definition** windows.



Each row in the grid defines a vertex or an arc segment. Vertices have **X** and **Y** coordinates that are located in relation to the center point of the pad, antipad, thermal pad, or hole. The center point is marked by a plus sign (**+**) on the diagram. Enter a distance on the center point for each coordinate. Select a row to find its corresponding point in the diagram.

Arc segments are curved segments defined by three points: a beginning, end, and apex. Each arc segment must have a vertex before and after it to define its beginning and end. Thus, arc

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

segments cannot be adjacent to one another. Each arc segments has an arc height or arc radius which can be changed in the grid.

The grid has a shortcut menu where you perform the following tasks. Different options appear depending on whether you select a row and whether you Right-click a vertex or arc segment.

- **Append Vertex** adds a new row for a vertex to the grid with the coordinates 0 mm, 0mm.
- **Insert Vertex Before** inserts a point before the current selection.
- **Insert Vertex After** inserts a point after the current selection.
- **Remove Vertex/Segment** removes the row you Right-clicked on.
- **Convert to Arc Segment** converts a vertex to an arc segment.
- **Insert Arc Segment Before** inserts an arc segment before the current selection.
- **Insert Arc Segment After** inserts an arc segment after the current selection.

The diagram displays a rendering of the polygon and the object's center point. Hover over a point to see its coordinates. Click a point to select it in the grid. Right-click the diagram to zoom in or out or to resize it to show the entire polygon. The diagram offers two options:

- **Fit On Change** redraws the diagram to fit the polygon whenever you make a change.
- **Show Original** displays an outline of the original polygon to compare it with any changes.

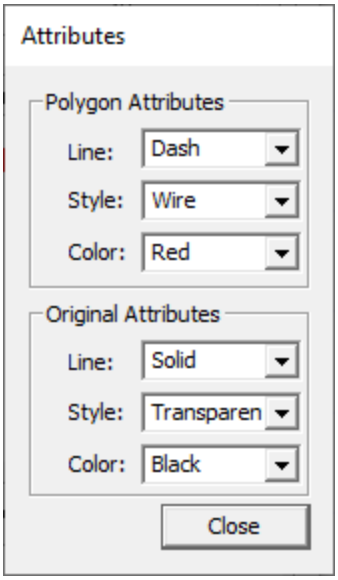
Edit the Polygon

1. Add new vertices as necessary.
    a. Highlight the row of the point just before where you want to add a vertex.
    b. Right-click and choose **Insert Vertex After**. A new row appears in the grid with the coordinates 0 mm, 0mm.
    c. Enter coordinates in the **X** and **Y** columns of the new row.
2. Add new arc segments as necessary.
    a. Highlight the row of the point just before where you want to add an arc segment.
    b. Right-click and choose **Insert Arc Segment After**. A new row appears in the grid.
    c. Above the grid choose whether you want to work with **Arc Heights** or **Arc Radii**.
    d. Enter the arc height or arc radius of the new segment in the new row.
3. Remove unnecessary vertices or segments.
    a. Highlight the row of the unwanted vertex or segment.
    b. Right-click and click **Remove Vertex/Segment**.
4. Check your work in the diagram.
5. Click **OK** to close the **Polygon Editor**. The polygon appears in the **Top view** diagram of the **Edit Padstack Definition** window. The **Cross section view** diagram is unchanged.

Change how the diagram looks

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 diagram and click **Show/hide attributes** to open an **Attributes** window. Here customize the lines, fill, and color of the polygon. Controls are available for both the edited polygon and the original polygon if **Show Original** is enabled.



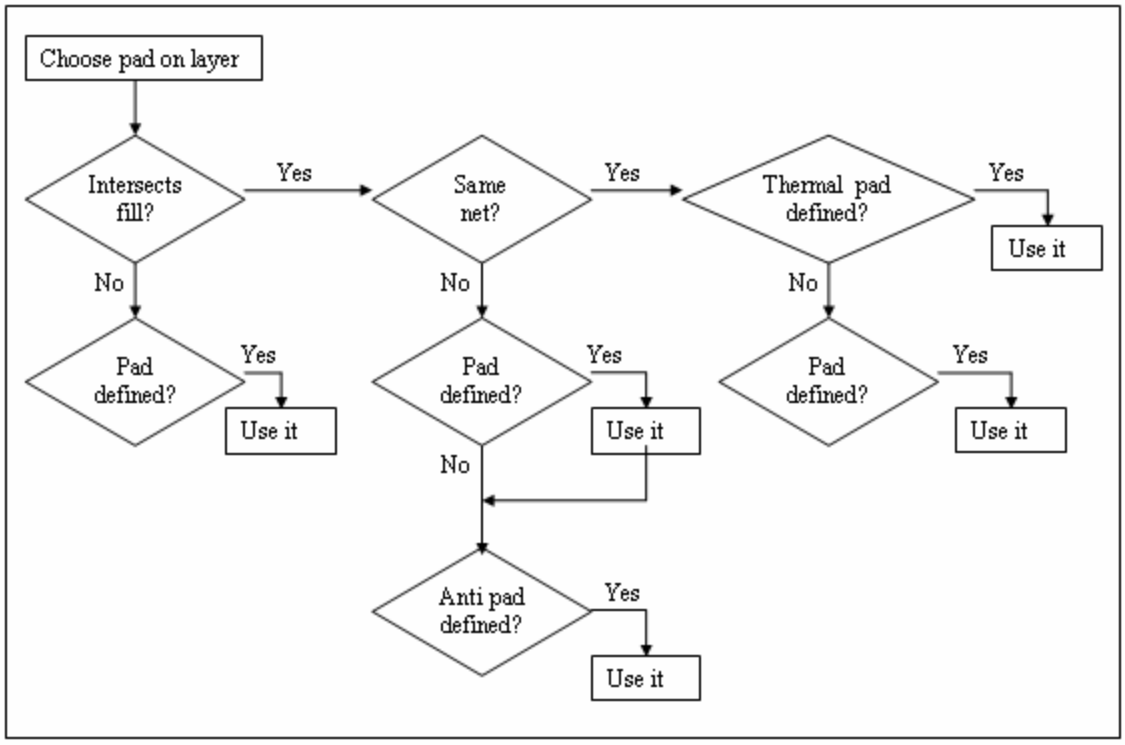
Pad Behavior in Vias and Pins

The pad(s) used in a particular via or pin are determined by the padstack definition and by the intersected primitives in their nets. Multiple pads are sometimes defined for the same layer.

- The intersection point between the central axis of a via or pin and the intersected layers is inspected to determine its metal type (fill, trace, or un-present). By definition, ground layers are fill-metal with primitives defining scratch areas.
- If a fill is intersected, the net of the fill is also taken into consideration.

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 the intersection of the fill/net/pads that are defined in the padstack definition, the choice of pad that is used is determined using the following algorithm:



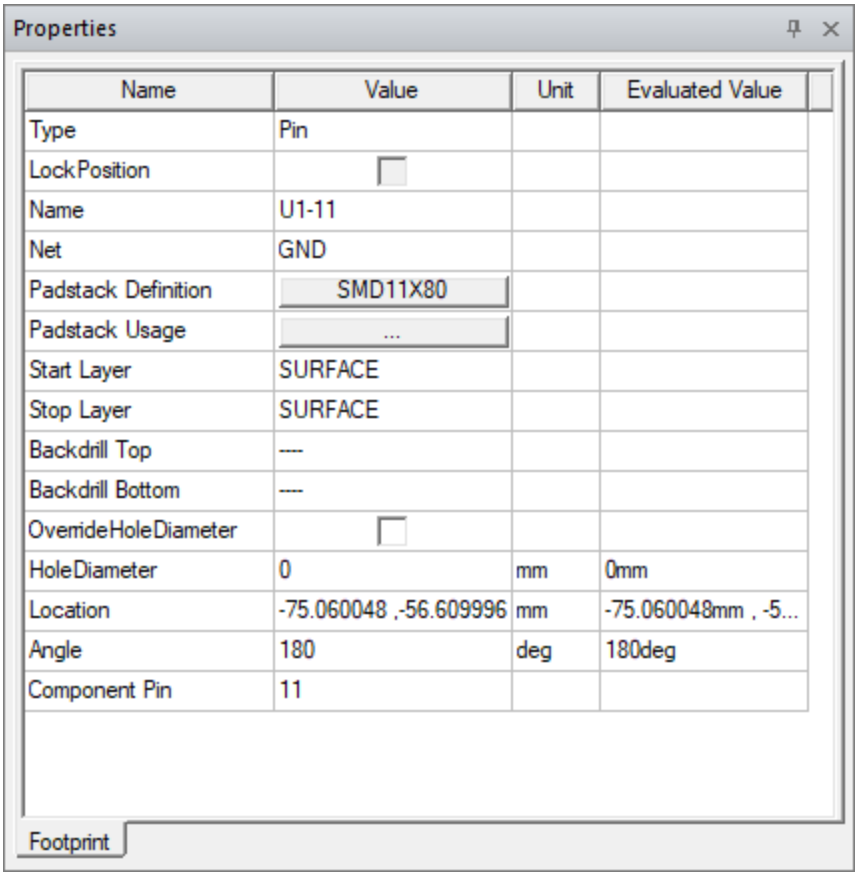
- The pad choice on the algorithm is subject to suppressing of non-functional internal pads as described in "Layout Editor Options: Object Panel " on page 7-11.

> **Note:**
>
> Anti pads subtract material on the fill area.

Editing Via and Pin Padstacks in Layout

To edit a via or pin padstack, click the via or pin to display its properties 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.



- **Name** is the name of the object.
- **Net** lists the net to which the object is assigned.
- **Override Hole Diameter** and **Hole Diameter** are used to override the diameter of the padstack definition for this particular via or pin.
- **Padstack Definition** opens the Edit Libraries window, which allows you to switch the padstack definition that is used by the via or pin.
- **Padstack Usage** opens the **Padstack Usage and Definition** window, to edit the padstack. When opened on the Properties window, information related to the padstack definition is highlighted in blue.
- **Type** determines whether the object definition is a via or pin.

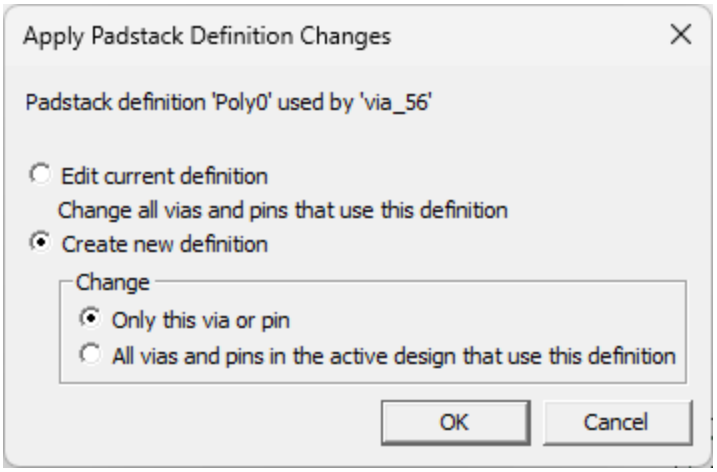
Initial Padstack Definition Layers

Special padstack definition layer names (**Start**, **Stop**, and **Default**) are provided to allow you some control over default mapping of the definition layers to stackup layers when the padstack is placed into a layout as a via or pin. Using these special layer names is optional. When a via or pin is placed, a default mapping occurs in the following order:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Any layout stackup signal layer before the start layer of the padstack or after the stop layer of the padstack does not get mapped to any definition layer.
2. Any layout stackup signal layer whose name matches a definition layer name is mapped to that definition layer.
3. If the start layer of the padstack is not yet mapped, and there is a definition layer named "Start", they are mapped.
4. If the stop layer of the padstack is not yet mapped, and there is a definition layer named "Stop", they are mapped.
5. If there is a definition layer named "Default", any remaining layout stackup signal layers are mapped to it.
6. If there are remaining layout stackup signal layers, they are mapped (in order) to the remaining definition layers.
7. Any layout stackup signal layers that are unmapped, remain that way.

When the **Padstack Usage and Definition** window is closed by clicking **OK**, you are given a choice of how to apply the definition changes that are made; the **Apply Padstack Definition Changes** window opens only if changes are made to the padstack definition (not all changes to the **Padstack Usage and Definition** window change the padstack definition).



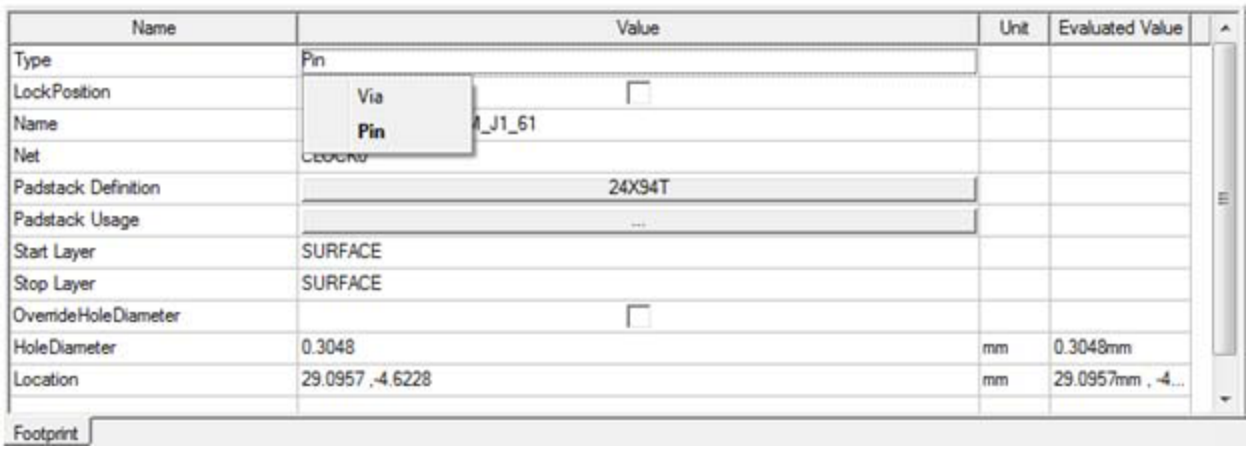
- **Edit current definition** — changes all vias and pins that use this definition
- **Create new definition** – allows you select whether the changes apply to **Only this via or pin** or whether the changes apply to **All vias and pins in the active design that use this definition** of the original padstack.

Pin and Net Padstack Operations

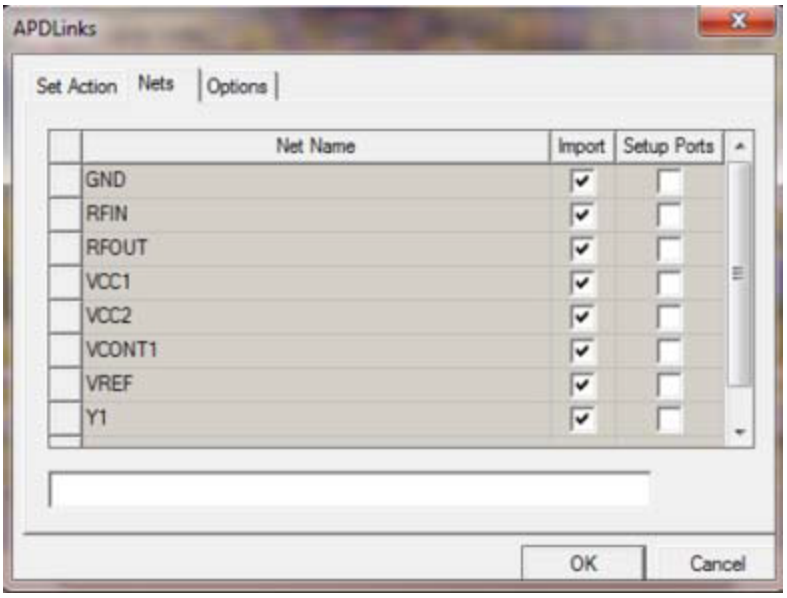
Padstack instances are pins or vias; either type may have an associated port. The "pin" designation does not imply that a port must be associated with the padstack. When a port is

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

removed from a pin, the object retains its "pin" type. The type is changed through the "Type" property:



Both APDLinks and ANFV2 imports correctly retain the "pin" and "via" type for padstacks. For APDLinks, only those nets for which "Setup Port" has been selected, has associate ports with the pins.



The following commands act on pins and nets. These are accessed through the shortcut menu on the Layout tab under "Nets" and also on the right-click layout menu. See Pin and Net Padstack Operations for information about the following scripts.

AddPortsToNet

Use:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

— Add ports to all the pins on the designated nets.

Command:

— In Layout, **Right-click > Port > Create Ports on Net**

— Layout tab under Nets, **Right-click > Create Ports**

RemovePortsFromNet

Use:

— Removes ports from all the pins on the designated nets.

Command:

— In Layout, **Right-click > Port > Remove Ports from Net**

— Layout tab under Nets, **Right-click > Remove Ports**

AddPortsToAllNets

Use:

— Adds ports to all the pins in all the nets.

Command:

— Layout tab under Nets, **Right-click > Create Ports**

RemovePortsFromAllNets

Use:

— Removes ports from all the pins in all the nets.

Command:

— Layout tab under Nets, **Right-click > Remove Ports**

Using the Estimate Module in HFSS 3D Layout

To access the Estimate module, click **Estimate** on the **HFSS 3D Layout** menu. Use the Estimate module to estimate parameters related to the following:

- Microstrip line
- Quarter-wave transformer
- Tuning stub
- Rectangular patch
- Circularly polarized (CP) patch

The menus in the Estimate module have the following functions:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T-Line	Estimates transmission-line quantities such as the characteristic impedance of a line, the line width corresponding to a characteristic impedance, the guided wavelength and the effective relative permittivity of a microstrip line or stripline, and the losses associated with a line.
Impedance Match	Estimates the dimensions of a quarter-wave section to match a load impedance to an input impedance. Also estimates the location and dimension of a tuning stub for matching an input impedance to a complex load impedance.
Printed Antennas	Estimates the dimensions of patch antennas.
Microwave Calculator	Converts between the mismatch loss, the reflection coefficient, the return loss, and the voltage standing wave ratio (VSWR). Also converts between dB and the ratio for both power and voltage values.
Units	Sets the current working units.

Estimating Transmission Line Quantities

Use the Estimate module's **T-Line** commands to estimate transmission line quantities.

The following commands are available:

Impedance	Estimates the characteristic impedance corresponding to a particular line width.
Width	Estimates the microstrip line width corresponding to a particular characteristic impedance.
Guided Wave Length	Estimates the guided wavelength and the effective relative permittivity of a microstrip line.
Line Loss	Estimates the metallic loss and dielectric loss of a microstrip line at a given frequency.

Estimating Characteristic Impedance

Use this command to estimate the characteristic impedance of a microstrip or stripline on the width you specify.

To estimate the characteristic impedance:

1. Select the signal layer on which the microstrip line lies, from the list.

   If the selected layer is an infinite negative signal layer, a CPW line is assumed.

2. Click **T-Line>Impedance**.

   Parameter input boxes appear in the right side of the Estimate module's 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. For trace layers, enter the width of the microstrip line in the **Width** box. The units for the line width are the same as the current working units.
4. For infinite negative signal layers, do the following:
   a. Type the width of the first line in the **Width1** area.

   The units for the line width are the same as the current working units.

   b. Type the width of the second line in the **Width2** area.
   c. Type the gap between the two lines in the **Gap** area.
   d. Type the frequency at which to estimate the impedance in the **Frequency** area.
5. Click **Calculate** to display the impedance underneath the data you entered.

The characteristic impedance corresponds to the line width you specified. The units for impedance is ohms.

Estimating Width

Use this command to estimate the width of a microstrip or stripline on the characteristic impedance you specify. This command is not applicable for a CPW line.

To estimate the width:

1. Select the trace layer on which the microstrip line lies.
2. Click **T-Line** > **Width**.

Estimate parameters and results boxes appear in the right side of the Estimate window.

3. Type the characteristic impedance, in ohms, of the microstrip line in the **Impedance** area.
4. Click **Calculate** to display the estimated width.

The line width corresponds to the specified impedance. The units for the width are the same as the current working units.

Estimating Guided Wavelength and Effective Relative Permittivity

You can estimate the guided wavelength and the effective relative permittivity of a stripline or microstrip line from a given width and frequency.

To estimate the guided wavelength and effective relative permittivity:

1. Select the signal layer on which the microstrip line lies.

If the selected layer is an infinite negative signal layer, a CPW line is assumed.

2. Click **T-Line** > **Guided Wavelength**.

Parameter input boxes appear on the right side of the Estimate module's 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. For trace layers, type the width of the microstrip line in the **Width** box in the current working units.
4. For infinite negative signal layers, do the following:
   a. Type the width of the first line in the **Width1** area.

      The units for the line width are the same as the current working units.

   b. Type the width of the second line in the **Width2** area.
   c. Type the gap between the two lines in the **Gap** area.
5. Type the frequency in the **Frequency** area, in GHz.

   This frequency is independent of the maximum frequency entered in the solution setup.

6. Click **Calculate** to perform the estimate.

   The estimated information appears in the following boxes:

Guided Wavelength	Displays the guided wavelength in the current working units.
Effective Relative Permittivity	Displays the effective relative permittivity.

The estimated guided wavelength and effective relative permittivity correspond to the line width and frequency you specified.

Estimating Line Loss

Use this command to estimate the line loss of a microstrip line on the width and frequency you specify. This command is not available for trace layers between two infinite negative signal layers.

To estimate the line loss:

1. Select the trace layer on which the microstrip line lies.
2. Click **T-Line** > **Line Loss**.

   A window opens on the right side of the Estimate module's window.

3. Type the width of the microstrip line in the **Width** area.

   The units for the width are the same as the current working units.

4. Type the frequency in the **Frequency** area, in GHz.

   This frequency is independent of the maximum frequency entered in the solution setup.

5. Click **Calculate** to perform the estimate.

   The estimated information appears in the following boxes:

Conductor	Displays the estimated loss due to finite conductivity and thickness 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.

Loss	metal.
Dielectric Loss	Displays the estimated line loss due to imperfect dielectric materials.
Total Loss	Displays the sum of the conductor loss and the dielectric loss of the line.

Estimating Matches Between Impedances

Use the **Impedance Match** menu to estimate matches between an input impedance and a load impedance. The following commands are available:

Quarter-Wave	Estimates the dimensions of a quarter-wave section to match a load impedance to an input impedance.
Tuning Stub	Estimates the location and the dimension of a tuning stub for matching an input impedance to a complex load impedance.

The **Quarter-Wave** transformer and the **Tuning Stub** commands offer two different ways of matching an input impedance to a load impedance. While quarter-wave transformers are more commonly used, tuning stubs may occupy less physical space. Tuning stubs can also be used to match a real input impedance to a complex load impedance, whereas quarter-wave transformers are used only with real input and load impedances.

> **Note:**
>
> The **Impedance Match** commands are only available for trace layers.

Estimating Quarter-Wave Transformer Dimensions

Click the **Impedance Match** > **Quarter-Wave** command to estimate the dimensions of a quarter-wave section to match a load impedance to an input impedance.

Before using this command, keep the following in mind:

- The numbers computed by this command are for the selected layer and are rough estimates only. Depending on the substrate parameters, the dimensions of the quarter-wave transformer most likely are different for metals on different layers; therefore, it is important that you select the appropriate signal layer first.
- The values estimated by this command may not agree exactly with the values computed by the simulation engine. This command is meant to help you obtain a first iteration design. If you are designing a quarter-wave transformer, you can use the rough dimensions provided as a starting point. Once you have these, you can draw and simulate the structure, then fine-tune the design.

To estimate a quarter-wave transformer:

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 layer on which the quarter-wave transformer lies.
2. Click **Impedance Match** > **Quarter-Wave**. Estimate parameters appear in the right side of the Estimate window.
3. Type the selected load impedance in the **Load Impedance** area, in ohms. The number you enter must be real and greater than 0.
4. Type the input impedance in the **Input Impedance** area. The number you enter must be real and greater than 0. The unit for impedance is ohms.
5. Type the frequency in the **Frequency** area, in GHz. This frequency is independent of the maximum frequency entered in the solution setup.
6. Click **Calculate** to perform the estimate. The estimated information appears in the following boxes:

Impedance	Displays the impedance required to match the input impedance to the load impedance. The unit for impedance is ohms.
Width	Displays the line width of the quarter-wave section in the current working units.
Length	Displays the line length of the quarter-wave section in the current working units.

Estimating Tuning Stub Parameters

Click the **Impedance Match** > **Tuning Stub** command to obtain an estimate of the location and the dimension of a tuning stub for matching an input impedance to a complex load impedance. The tuning stub is assumed open-ended. If you have a microstrip line terminated in a complex load impedance, you can also use this command to estimate the location along the microstrip line where the input impedance, looking toward the load, is purely resistive.

Before using this command, keep the following things in mind:

- The numbers computed by this command are rough estimates only. Depending on the substrate parameters, the patch parameters may be different for patches on different layers; therefore, it is important that you select the appropriate signal layer before accessing this command.
- The estimated values may not agree exactly with the values computed by the simulation program. This command is meant to help you obtain a first iteration design. If you are designing a tuning stub, you can use the rough dimensions provided as a starting point. Once you have these, you can draw and simulate the structure, then fine-tune the design.

To estimate the location and the dimensions of a tuning stub:

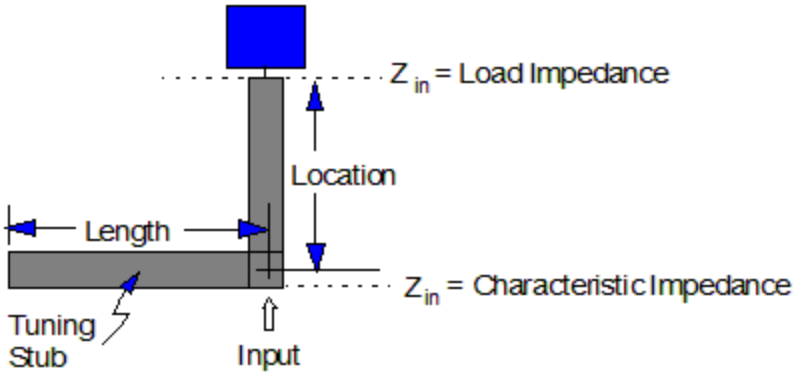
1. Select the trace layer on which the tuning stub lies.
2. Click **Impedance Match** > **Tuning Stub**. Estimate parameters appear on the right side of the Estimate window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type the frequency in the **Frequency** area, in GHz. This frequency is independent of the maximum frequency entered in the solution setup.
4. Type the line width of the microstrip line connected to the load impedance in the **Line Width** area. The units for the width are the same as the current working units. The width of the tuning stub is assumed the same as the width of the line connected to the load.
5. Type the real part of the complex load impedance in the **Real** area. The number you enter must be real and greater than 0. The units for impedance are ohms.
6. Type the imaginary part of the complex load impedance in the **Imaginary** box.
7. Click **Calculate** to perform the estimate. The estimated information appears in the following boxes:

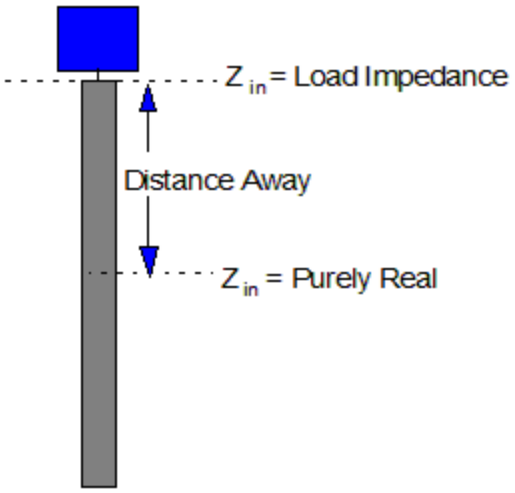
Location, D	Displays the estimated location for the tuning stub in the current working units. This is the distance measured on the load impedance along the microstrip line.
Length, L	Displays the estimated line length for the open-ended stub in the current working units. The width of the stub is assumed the same as width of the microstrip line specified in the input area.
Distance Away	Displays the location along the microstrip line where the input impedance, looking toward the load, is purely real. The estimated location is the distance measured on the load along the microstrip line. The unit for the distance is the same as the current working units.
Real Impedance	Displays the impedance value at the location along the microstrip line where the input impedance, looking toward the load, is purely real.

The following figure displays the location and line length for an open-ended stub. The width of the stub is assumed the same as the width of the microstrip line specified in the input area:



The following figure displays the location along a microstrip line where the input impedance is purely real:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Estimating Dimensions of Printed Antennas

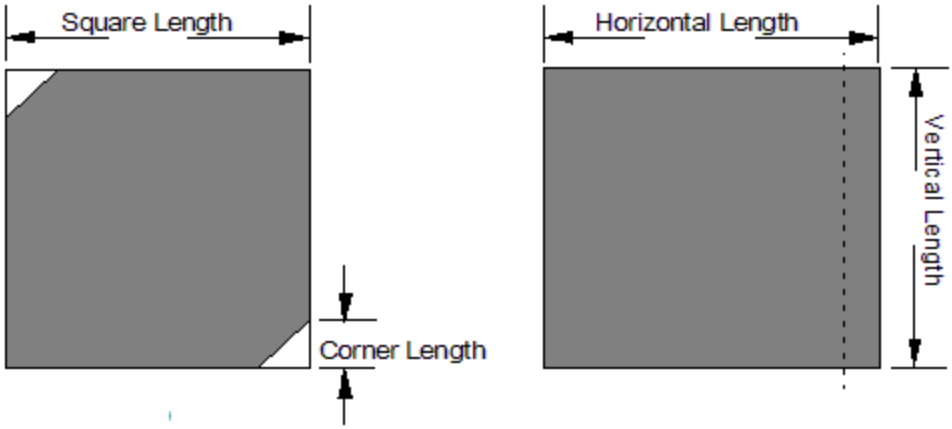
Use the **Printed Antennas** menu to estimate the dimensions of two different types of singly fed, circularly polarized (CP) patches and the resonating length of a rectangular patch. This command is not available for infinite negative signal layers or trace layers between two infinite negative signal layers.

The following commands are available:

CP Corners Truncated	Estimates the dimensions of a corners-truncated CP element.
CP Nearly Square	Estimates the dimensions of a nearly square CP element.
Rectangular Patch	Estimates the resonating length of a rectangular patch. This command also estimates the bandwidth and the radiation efficiency of the patch and the location on the patch corresponding to an impedance value.

Click the **CP Corners Truncated** and **CP Nearly Square** commands to obtain estimates on the dimensions of two different types of singly fed CP patches. To generate circular polarization on a patch using a single feed, you must excite two orthogonal modes on the patch. To obtain two orthogonal modes on a patch, you generally start with a square patch and perturb the element. You can perturb the element by cutting away two opposite corners of the patch or by extending one of the sides slightly. The following figure demonstrates the two types of singly fed CP patches:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Estimating Corners-Truncated CP Element Parameters

Use this command to estimate the dimensions of a corners-truncated, circularly polarized (CP) element. The feed points for these patches are not arbitrary. For a corners-truncated patch, you need to feed the patch along the center line of the patch in order to excite the two orthogonal modes.

Before using this command, keep the following in mind:

- The values estimated by this command are for the current active layer only. Depending on the substrate parameters, the patch parameters may be different for patches on different layers; therefore, it is important that you select the appropriate signal layer before accessing this command.
- This command is unavailable if your structure does not contain negative signal planes or if it contains two negative signal planes and you have selected a signal layer between them.
- The method used to estimate the patch parameters is an approximate method based on the cavity model; therefore, the values given are only rough estimates.

To estimate a CP corners-truncated patch:

1. Select the trace layer on which the patch lies.
2. Click **Printed Antennas** > **CP Corners Truncated**. Estimate parameters appear on the right side of the Estimate module's window.
3. Enter the frequency in the **Frequency** area, in GHz. This frequency is independent of the maximum frequency entered in the solution setup.
4. Click **Calculate** to perform the estimate. The estimated information appears in the following boxes:

   Square Length Displays the estimated dimension of the unperturbed square patch in the current working units.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Corner Length	Displays the estimated dimension of the corner to cut away on the unperturbed square patch in the current working units. The shape of the cut-away corner is a right angle triangle having two equal sides along the edges of the patch.

Estimating Nearly Square CP Element Parameters

Use this command to estimate the dimensions of a nearly square, circularly polarized (CP) element. The feed points for these patches are not arbitrary. For a nearly square patch, you need to feed the patch at the corner of the patch.

Before using this command, keep the following things in mind:

- The values estimated by this command are for the current active layer only. Depending on the substrate parameters, the patch parameters may be different for patches on different layers; therefore, it is important that you are on the selected signal layer before accessing this command.
- This command is unavailable if your structure does not contain negative signal planes or if it contains two negative signal planes and you have selected a signal layer between them.
- The method used to estimate the patch parameters is an approximate method based on the cavity model; therefore, the values given are only rough estimates.

To estimate a CP nearly square patch:

1. Select the trace layer on which the patch lies.
2. Click **Printed Antennas** > **CP Nearly Square**. Estimate parameters appear on the right side of the Estimate module's window.
3. Enter the frequency in the **Frequency** area, in GHz. This frequency is independent of the maximum frequency entered in the solution setup.
4. Click **Calculate** to perform the estimate. The estimated information appears in the following boxes:

Horizontal Length	Displays the dimension of the perturbed patch in the horizontal or x direction in the current working units.
Vertical Length	Displays the dimension of the perturbed patch in the vertical or y direction in the current working units.

Estimating Rectangular Patch Parameters

Use this command to obtain a rough estimate of the resonating length of a rectangular patch. This command also estimates the bandwidth and the radiation efficiency of the patch and the location on the patch that corresponds to a user-specified impedance value.

Before using this command, keep the following 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.

- The method used is an approximate method based on the cavity model. The numbers estimated by this command are only rough estimates. Depending on the layer parameters, the patch parameters may be different for patches on different layers; therefore, it is important that you select the appropriate trace layer before using this command.
- The estimated values may not agree exactly with the values computed by the simulation engine. The purpose of this command is to help you obtain a first iteration design. If you are designing a patch, you can use the rough dimensions as a starting point. With these, you can draw and simulate the structure, then fine-tune the design.
- This command is unavailable if your structure does not contain negative signal planes or if it contains two negative signal planes and you have selected a signal layer between them.

To estimate a rectangular patch:

1. Select the trace layer on which the patch lies.
2. Click **Printed Antennas** > **Rectangular Patch**. Estimate parameters appear on the right side of the Estimate module's window.
3. Select **Square Patch** to enable the square patch option. For square patches, you do not need to specify the non-resonating length of the patch, which is entered in the **Vertical Length, W** area.
4. Enter the frequency in the **Frequency** area, in GHz. This frequency is independent of the maximum frequency entered in the solution setup.
5. Enter the dimension for the non-resonating edge of the rectangular patch in the **Vertical Length, W** area. The number must be greater than 0. The units are the same as the current working units.

   For a rectangular patch, the resonating edge of the patch is along the horizontal, or x, direction and the non-resonating edge is along the vertical, or y, direction. This box is unavailable if the **Square Patch** option is selected.

6. Enter the selected voltage standing wave ratio (VSWR) in the **VVSWR** area. The number entered must be greater than or equal to 1. This value is used to estimate the bandwidth of the patch.
7. Enter the selected input impedance in the **Input Impedance** area, in ohms. The number you specify must be greater than 0. This option is useful for finding an optimal location for feeding or loading the patch. When the estimate is calculated, HFSS estimates a location on the patch where the input impedance is close to the value you specified.
8. Click **Calculate** to perform the estimate. The estimated information appears in the following boxes:

Horizontal Length, L	Displays the estimated length for the resonating edge of the patch antenna. The units are the same as the current working units.
Bandwidth (%)	Displays the estimated bandwidth (in percent) at the VSWR setting you specified for the patch antenna.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiation Efficiency (%)	Displays the estimated radiation efficiency (in percent) for the patch antenna.
Impedance Location	Displays the approximate location on the patch where the input impedance is close to the impedance you specified. This location is specified by a horizontal offset and a vertical offset. The **Horizontal Offset** and **Vertical Offset** group boxes display these offset values. The horizontal offset value is the distance along the x direction relative to the lower-left node of the patch. The vertical offset value is the distance in the y direction relative to the lower-left node of the patch. The unit for these boxes is the same as the current working unit.

Using the Microwave Calculator

Click the **Microwave Calculator** menu to convert between common values. The following commands are available:

Microwave Parameters	Convert between mismatch loss in dB, reflection coefficient, return loss in dB, and voltage standing wave ratio (VSWR).
dB/Ratio Exchange	Convert between dB and ratio for both power and voltage values.

Converting between Microwave Parameters

Use this command to convert between the following common values:

- Mismatch loss in dB
- Reflection coefficient
- Return loss in dB
- Voltage standing wave ratio (VSWR)

To convert between the preceding common microwave parameter values:

1. Select the microwave parameter you want to convert - **Mismatch Loss (dB)**, **Return Loss (dB)**, **Reflection Coefficient**, or **Standing Wave Ratio**. **Mismatch Loss (dB)** is selected by default.

   A value input box appears to the right, corresponding to the selected parameter.

2. Enter a value for the selected parameter in the value input box.
3. Click **Calculate** to calculate the equivalent values for the other three parameters. The following equations are used when converting.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$RL = 10^{-(RLdB/20)}$$

$$VSWR = \frac{1+RL}{1-RL}$$

$$MismatchLoss = 10\log(1-RL^2)$$

where

- *RL* is the return loss.
- *RLdB* is the return loss in dB.

The equivalent values for the other three parameters appear in the Estimate Results area of the Estimate module.

Converting Between dB and Ratio

Use this command to convert dB and ratio for both power and voltage values.

To convert between dB and its ratio:

1. Select the parameter to convert from — either **dB** or **Ratio**. A value input box appears to the right, corresponding to the selected parameter.
2. Enter a value for the parameter selected.
3. Click **Calculate**.
4. The calculated dB or ratio value appears in the Estimate Results area of the Estimate module.
5. Select the type of unit measurement, **Power** or **Voltage**, for the calculated value:

Power Uses the power value, such as gain, for calculating dB:

$$dB = 10\log(ratio)$$

Voltage Uses the voltage value, such as fields, for calculating dB:

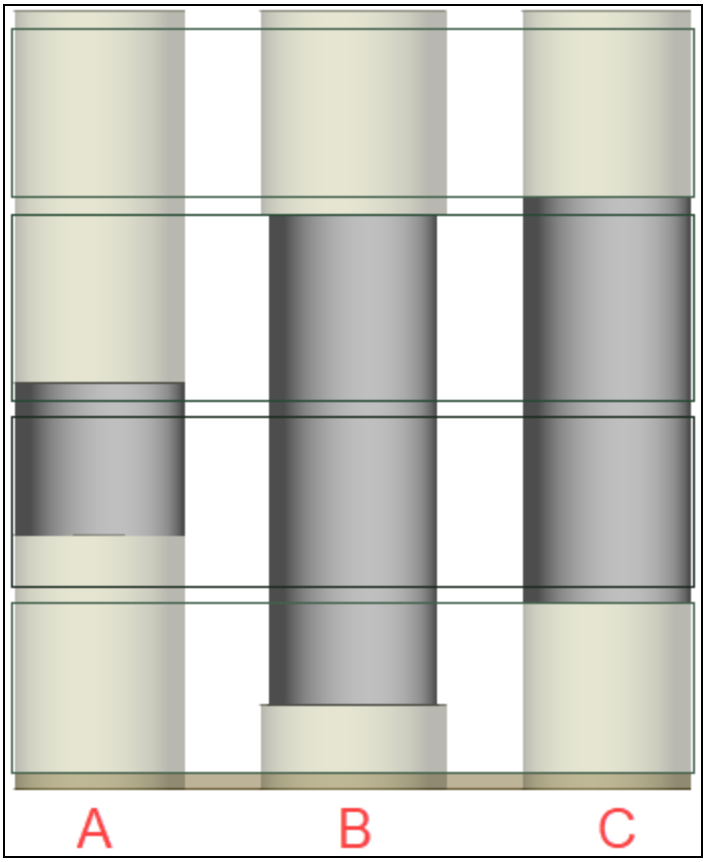
$$dB = 20\log(ratio)$$

The calculated results for the selected unit are shown in the **Estimate Results** area of the Estimate module.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backdrilling

Unused portions of vias (i.e., stubs) can create significant signal integrity disturbances. Backdrilling is a technique that removes these stubs. A backdrill can be defined in a combination of ways, as shown in the following screenshot.



A – **By Depth**: a user-specified length from above the top of the stackup, below the bottom of the stackup, or from both above the top and below the bottom of the stackup

B – **By Layer** with an offset (i.e., user-specified stub length): a user-selected stackup layer is selected on the **To Layer** drop-down menu. The backdrill will reach to the specified distance offset on the layer from above (i.e., when set in a **Top** subgroup box), or below (i.e., when set in a **Bottom** subgroup box).

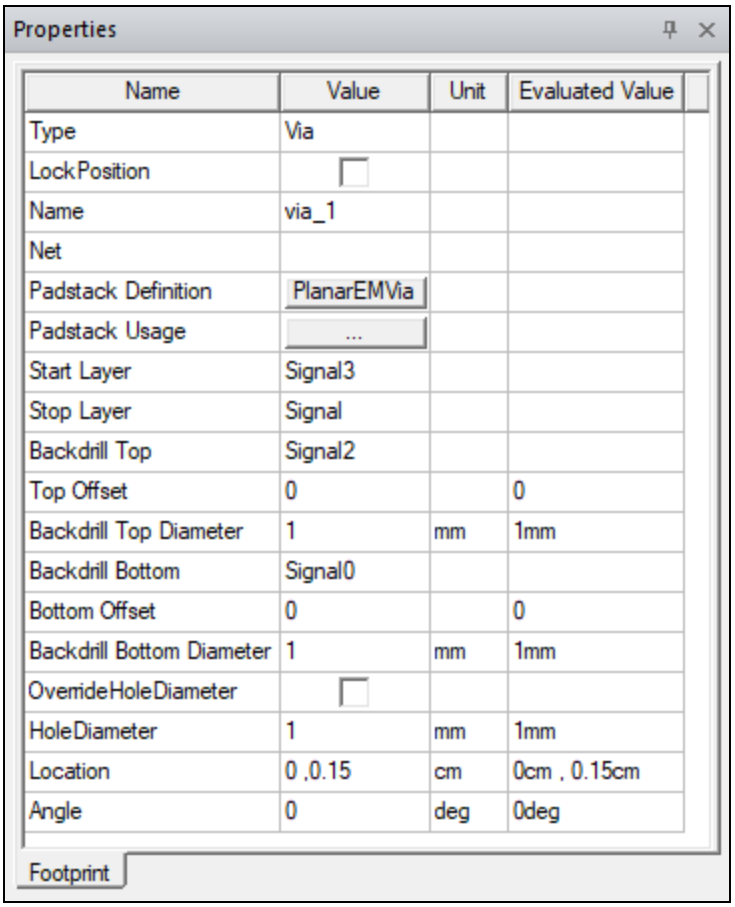
> **Note:**
>
> In this example, a layer and offset were selected on the **Top** subgroup box, but a user-specified depth is set in the **Bottom** subgroup box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C – **By Layer** (with no offset): a user-selected layer is selected from a drop-down menu. The backdrill will reach it from above (i.e., when set in a **Top** subgroup box), or below (i.e., when set in a **Bottom** subgroup box).

Viewing Backdrill Properties

A backdrill can be added to any 3D Layout design that includes a via, assuming the via's padstack definition has been configured (refer to Drawing a Via in the Layout Editor and Editing a Padstack). If a selected via has a backdrill, the **Properties** window will populate with a list of associated parameters.



Continue to Defining Backdrill Parameters.

Defining Backdrill Parameters

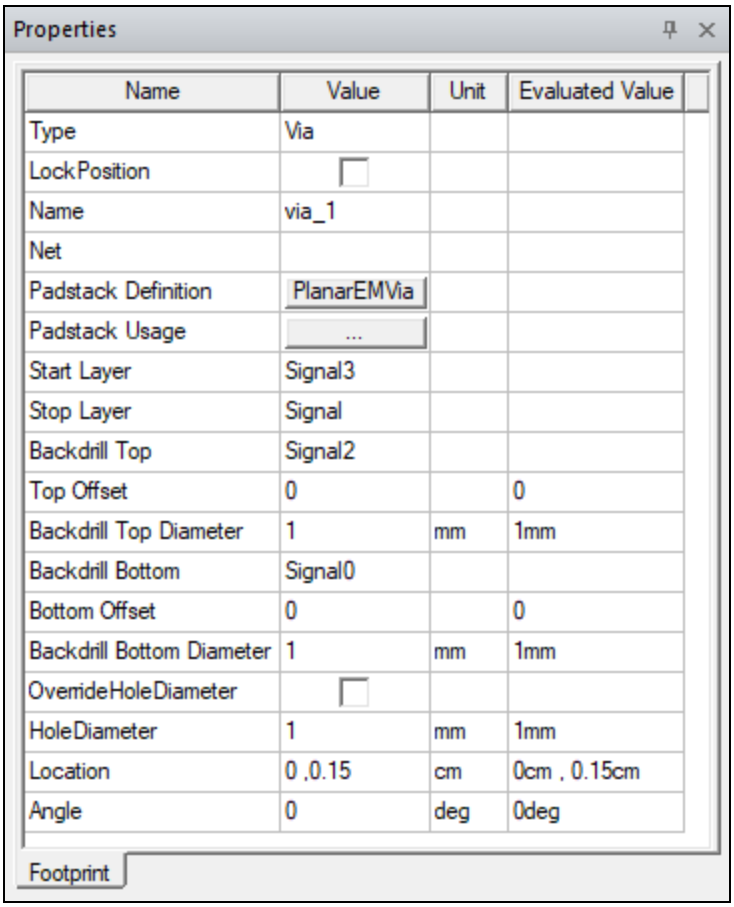
Complete the steps in the following sections to define backdrill parameters.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing Backdrill Parameters

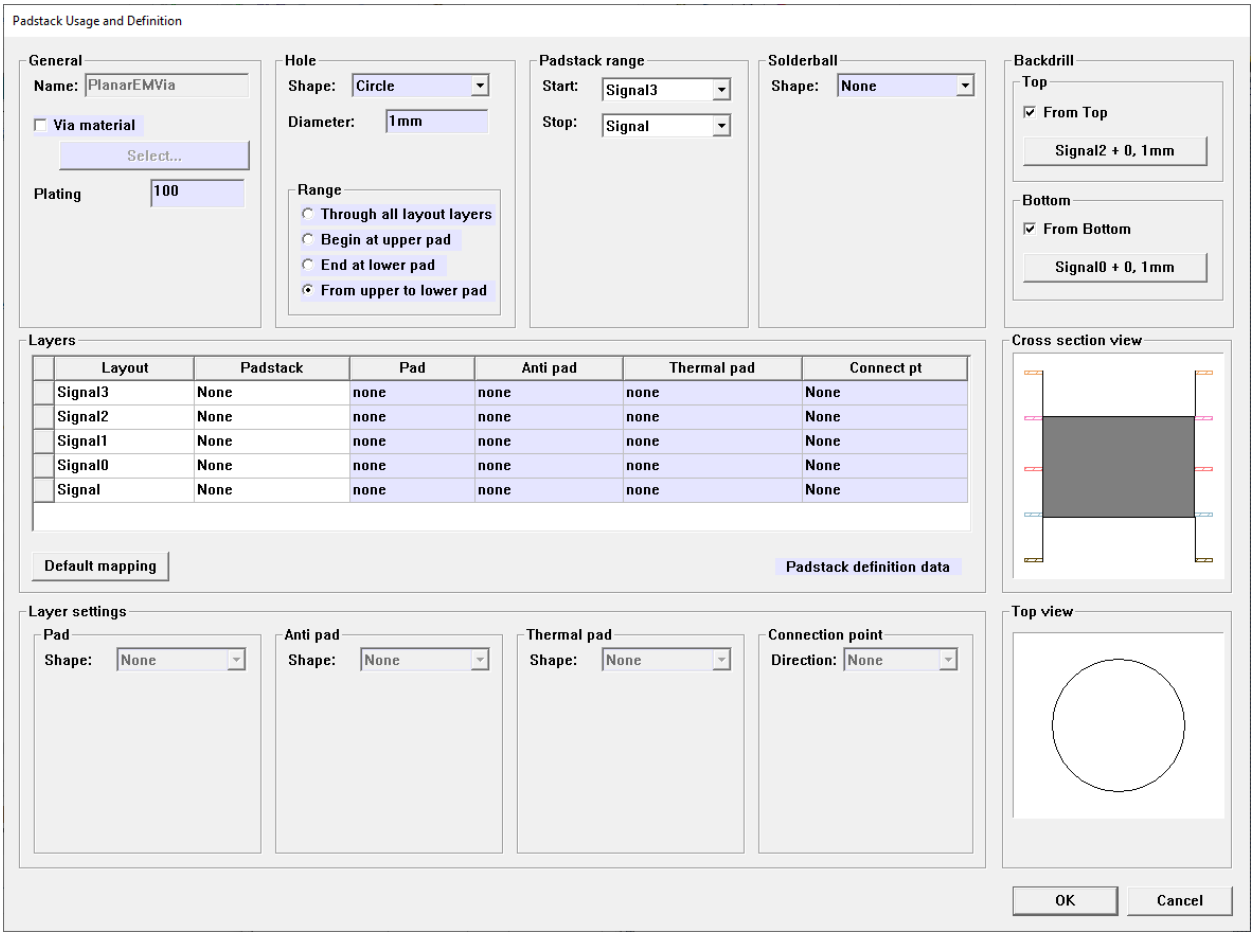
Follow these steps to view backdrill parameters.

1. From the **Layout Editor**, select a via to display its parameters in the **Properties** window.



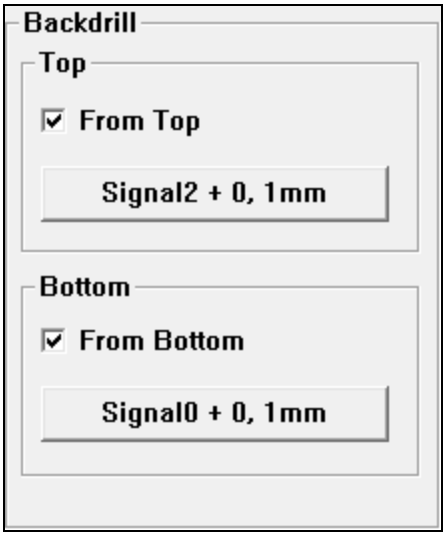
2. From the **Properties** window, click the **Padstack Usage [...]** button in the **Value** column to open the **Padstack Usage and Definition** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



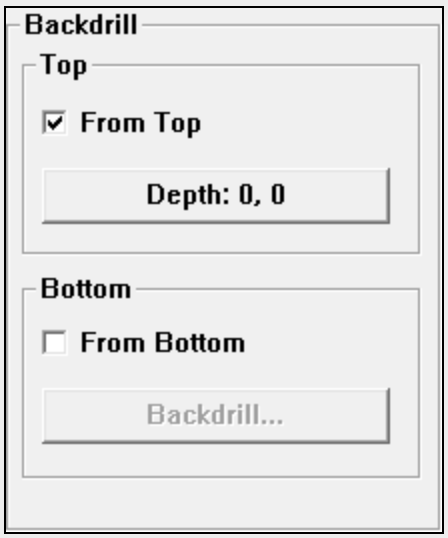
3. Current backdrill parameters are accessible on the **Backdrill** area in the upper-right corner.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

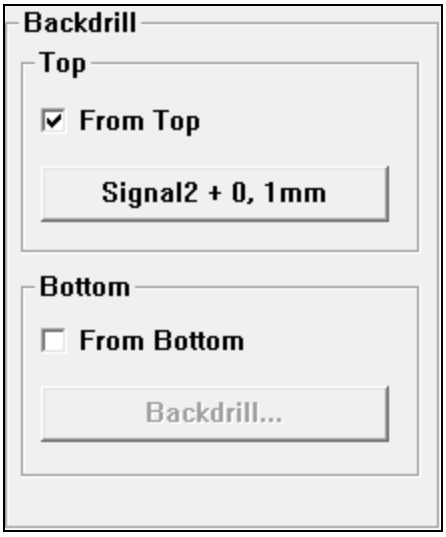
The Backdrill area is subdivided into two smaller **Top** and **Bottom** subgroup boxes. Setting the parameters in either area determines where the backdrill begins (i.e., **From Top** determines the depth of the backdrill on the top layer of the design and **From Bottom** determines the depth of the backdrill on the bottom layer of the design). The backdrill ends at the depth selected by users in the **Backdrill 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.

Activating/Deactivating Backdrill Subgroup Boxes

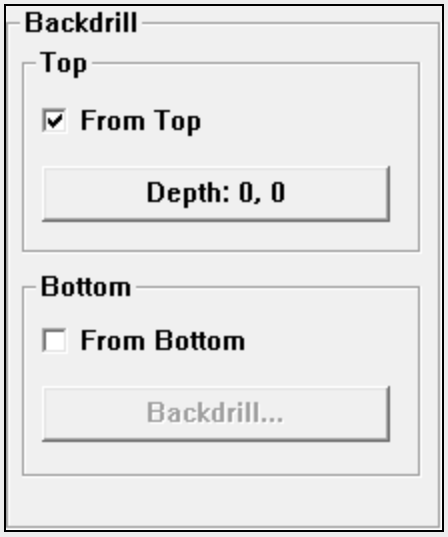
To activate or deactivate backdrilling on the top or bottom, add or remove checks from the **From Top** and/or **From Bottom** boxes.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Once a backdrill subgroup box is activated, its associated button displays an abbreviated description of the backdrill's current parameters (default is **Depth: 0, 0** before setup and **Backdrill** when deactivated). Once its parameters are defined in the **Backdrill Definition** window, the parameters are retained even if the subgroup box is deactivated. However, if the **Padstack Usage and Definition** window is closed once a backdrill subgroup is deactivated, the parameters are cleared.

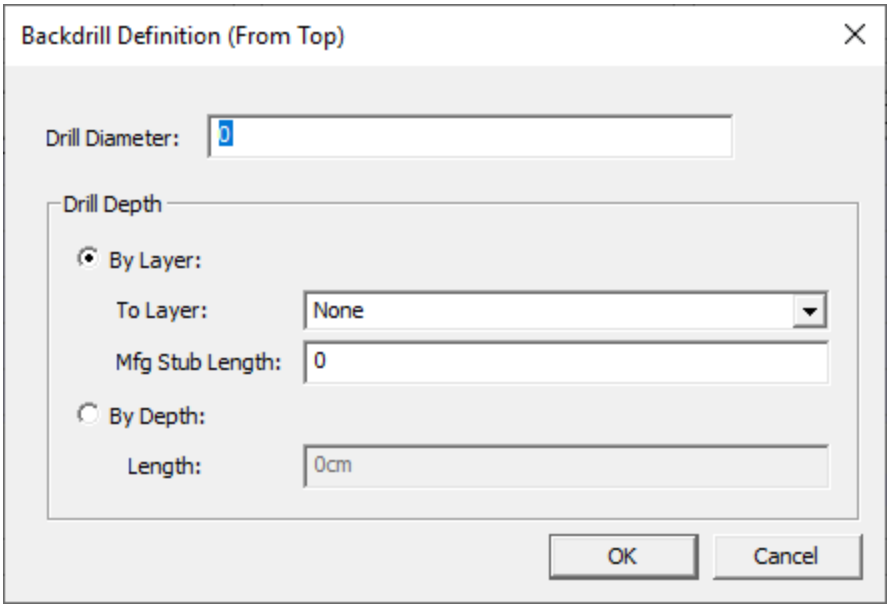


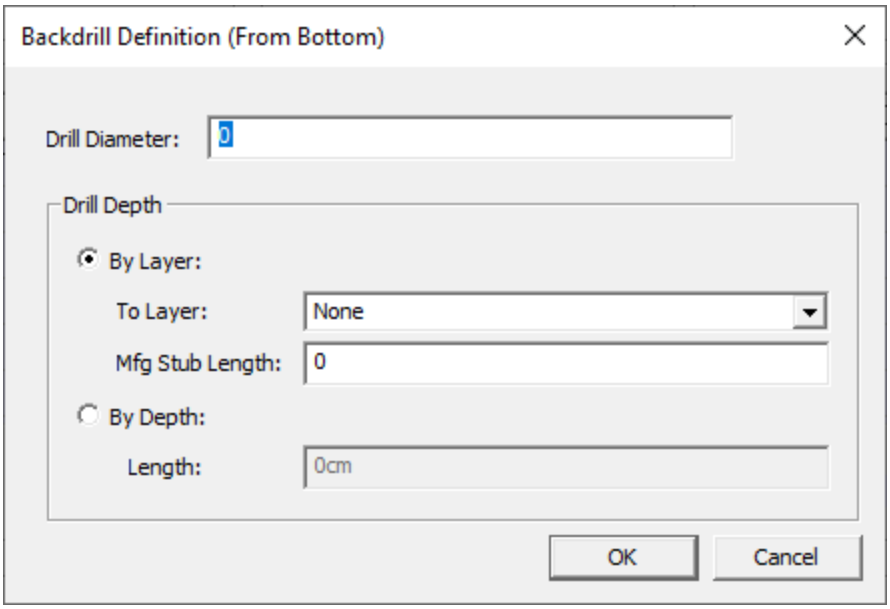
Setting Parameters in the Backdrill Definition Window

Follow these steps to configure a backdrill.

1. Once a subgroup box is activated, its associated button also activates. Click the button to open the **Backdrill 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.





2. Enter the following parameters to define the top or bottom backdrill.
    - **Drill Diameter** – enter the selected diameter of the backdrill

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 **Drill Depth** area, select **By Layer** or **By Depth**.

> **Note:**
>
> Aside from numerical values and units of measurement, project variables can be entered in the **Layer** drop-down menu, **Mfg Stub Length** (i.e., offset) field, and in the **Depth Length** field. Doing so offers the benefit of making changes to the variables in the **Properties** window and seeing the effect on the design in the **Layout Editor** in real time. Refer to "Using Project Variables to Modify a Backdrill" on page 11-119.

If **By Layer** is selected:

- **To Layer** – select a stackup layer selected from a drop-down menu, which the backdrill will reach from above (i.e., when set in the **Top** window), or below (i.e., when set in the **Bottom** window)
- **Mfg Stub Length** – enter the selected offset on the selected layer (i.e., when set in the **Top** window), or below the selected layer (i.e., when set in the **Bottom** window)

If **By Depth** is selected:

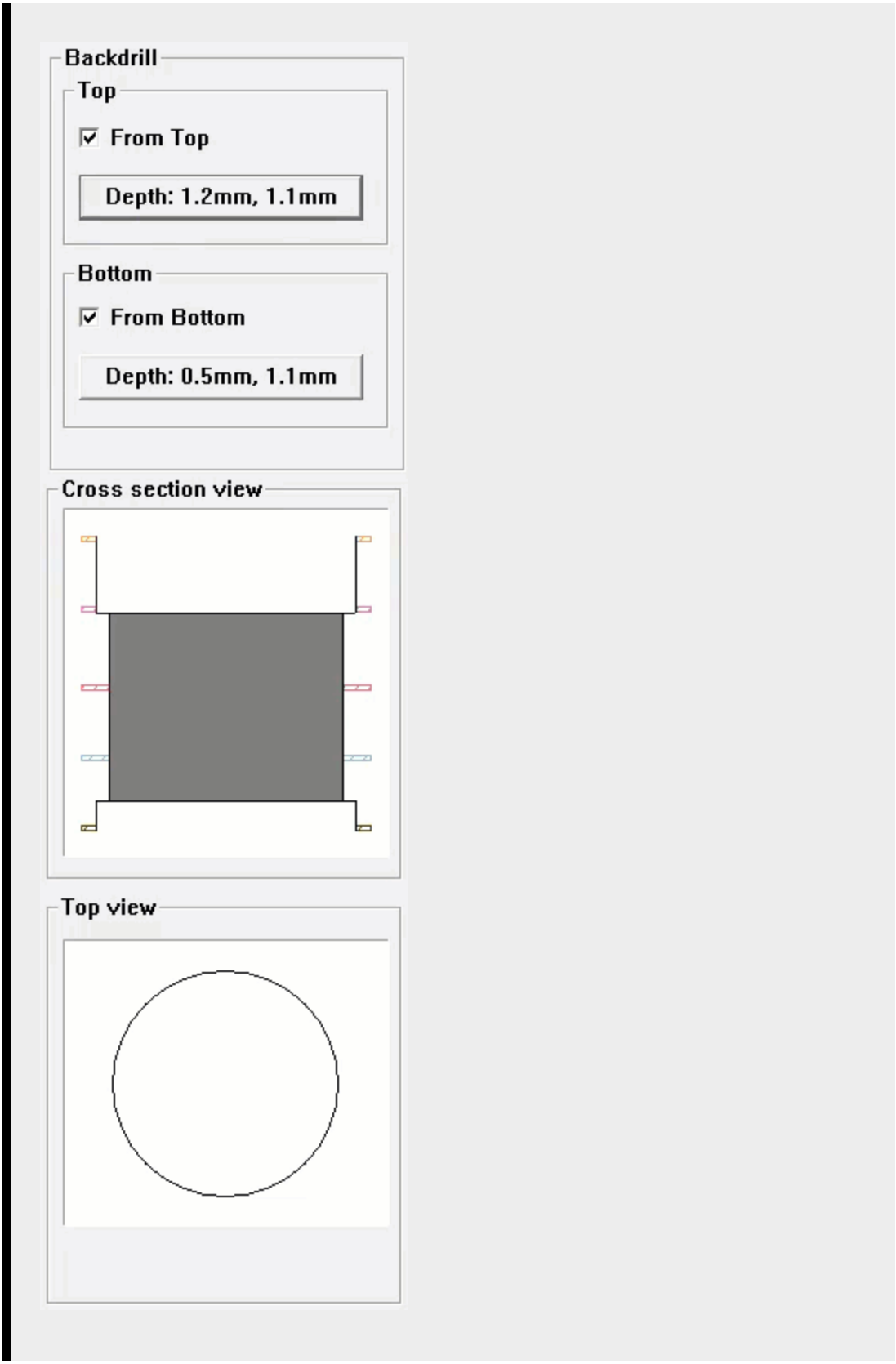
- **Length** – enter the selected length the backdrill should reach from above the top of the stackup (i.e., when set in the **Top**window), below the bottom of the stackup (i.e., when set in the **Bottom** window), or from both above the top and below the bottom of the stackup

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 parameters are set, the representative model shown in the **Cross section view** and **Top view** group boxes at the **Padstack Usage and Definition** window will change in real time. These group boxes display an approximation of the selected via's current backdrill definition for quick reference. Backdrills are rendered as empty space.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



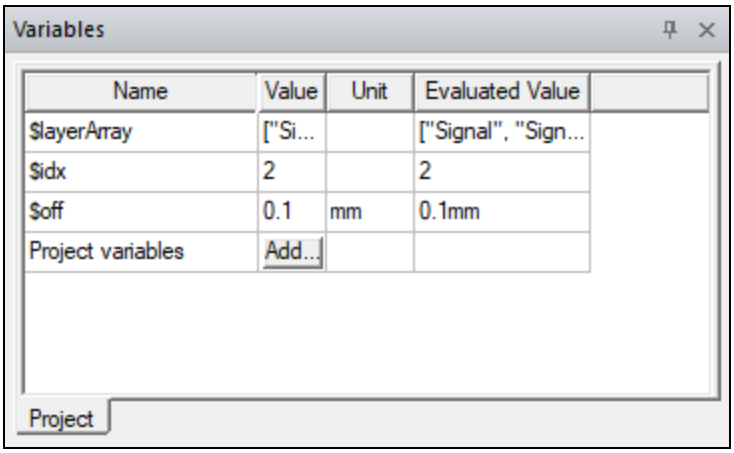
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Once the backdrill is defined, click **OK** to close the **Backdrill Definition** window.

Using Project Variables to Modify a Backdrill

Aside from numerical values and units of measurement, project variables can also be entered in the **Drill Depth** area fields to define a backdrill. From the following example, variables are used to define a backdrill and then the definition is modified on the **Properties** window. The effects appearing in the **Layout Editor**.

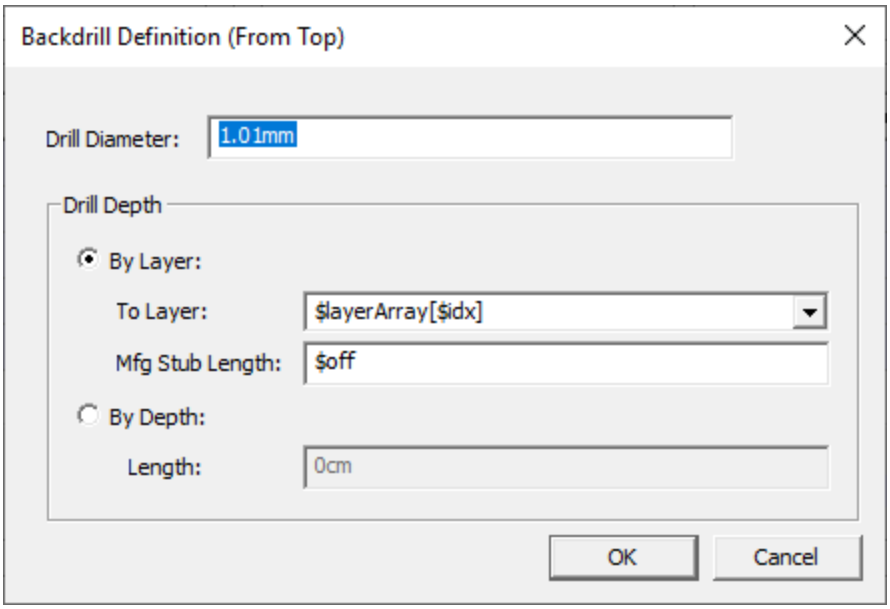
1. In this example, the **Variables** window is populated with a layer array variable with designations for each layer in the padstack, an index variable to associate a number with each layer, and an offset variable. Refer to Working with Variables for more information about creating project variables, design variables, and arrays.



2. Define a backdrill using project variables. In this example, the layer array variable and index variable are entered in the **To Layer** field and the offset variable is entered in the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

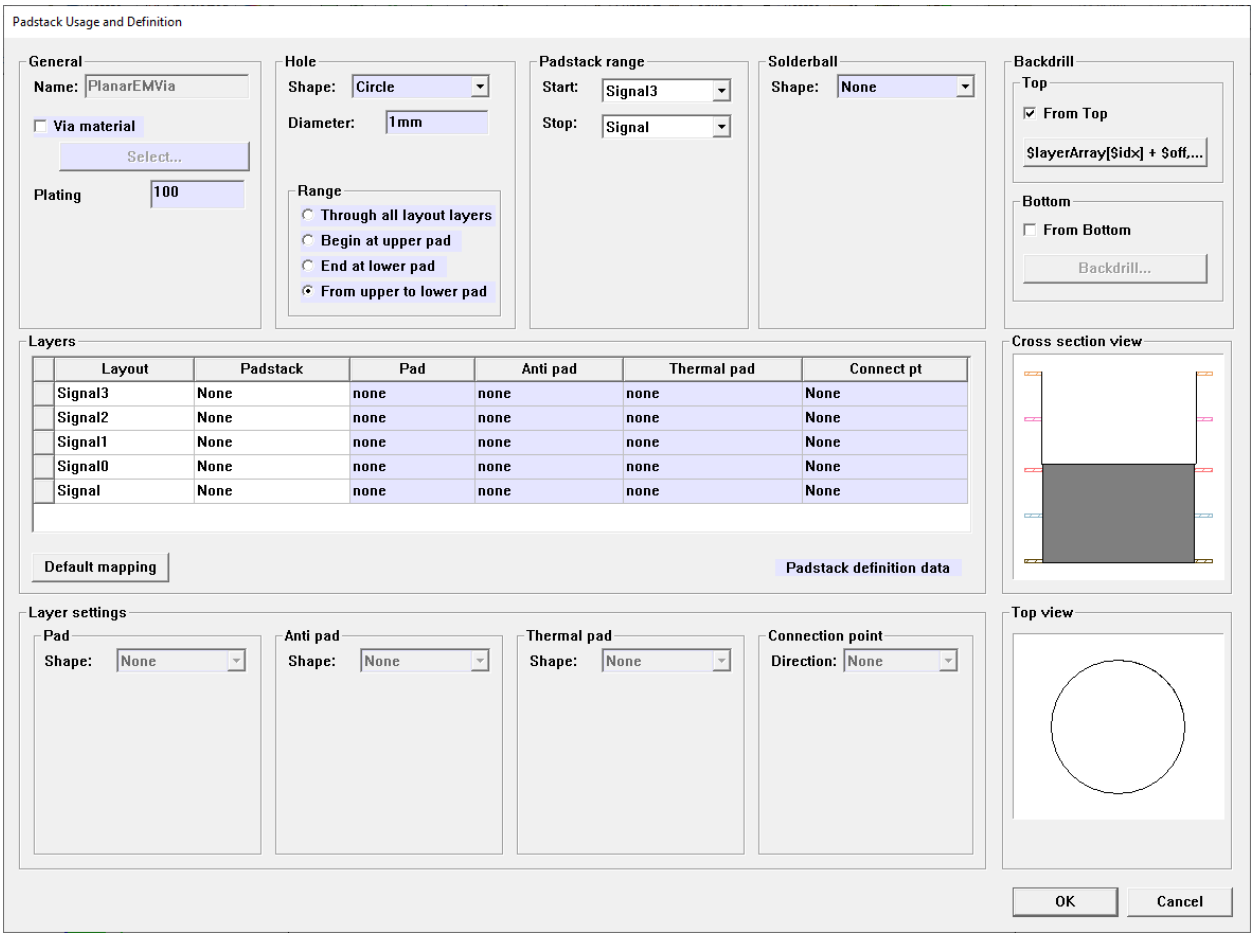
Mfg Stub Length field.



3. Click **OK** to close the **Backdrill Definition** window and return to the **Padstack Usage and Definition** window. The variables are abbreviated in the **Backdrill** area > **Top**

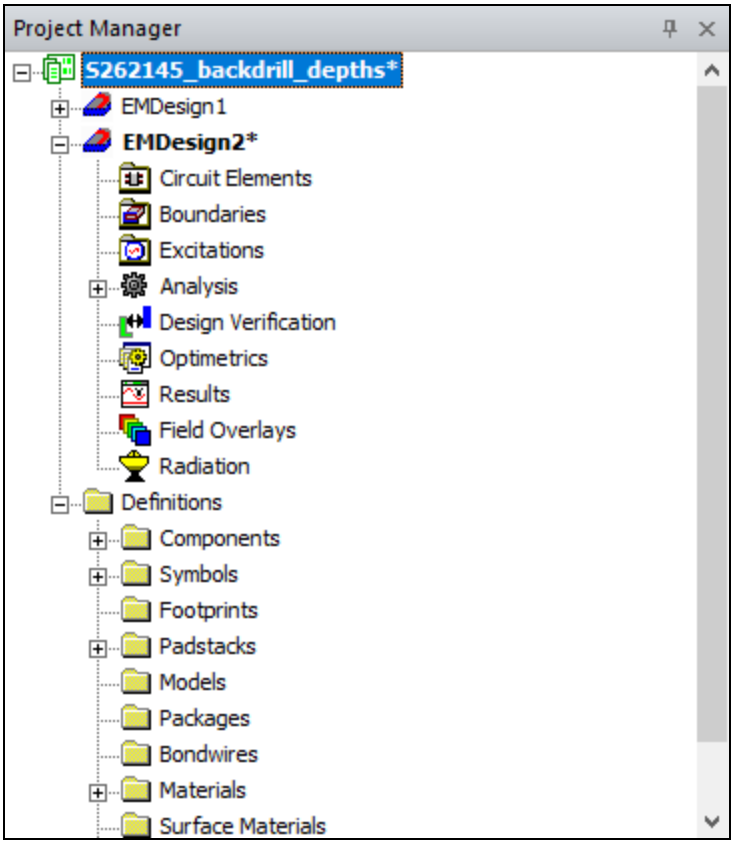
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

subgroup box button.



4. Cick **OK** to close the **Padstack Usage and Definition** window and return to the **Layout Editor**.
5. From the **Project Manager** window, select the **Project Tree** to populate the **Properties** window with the designated project variables.

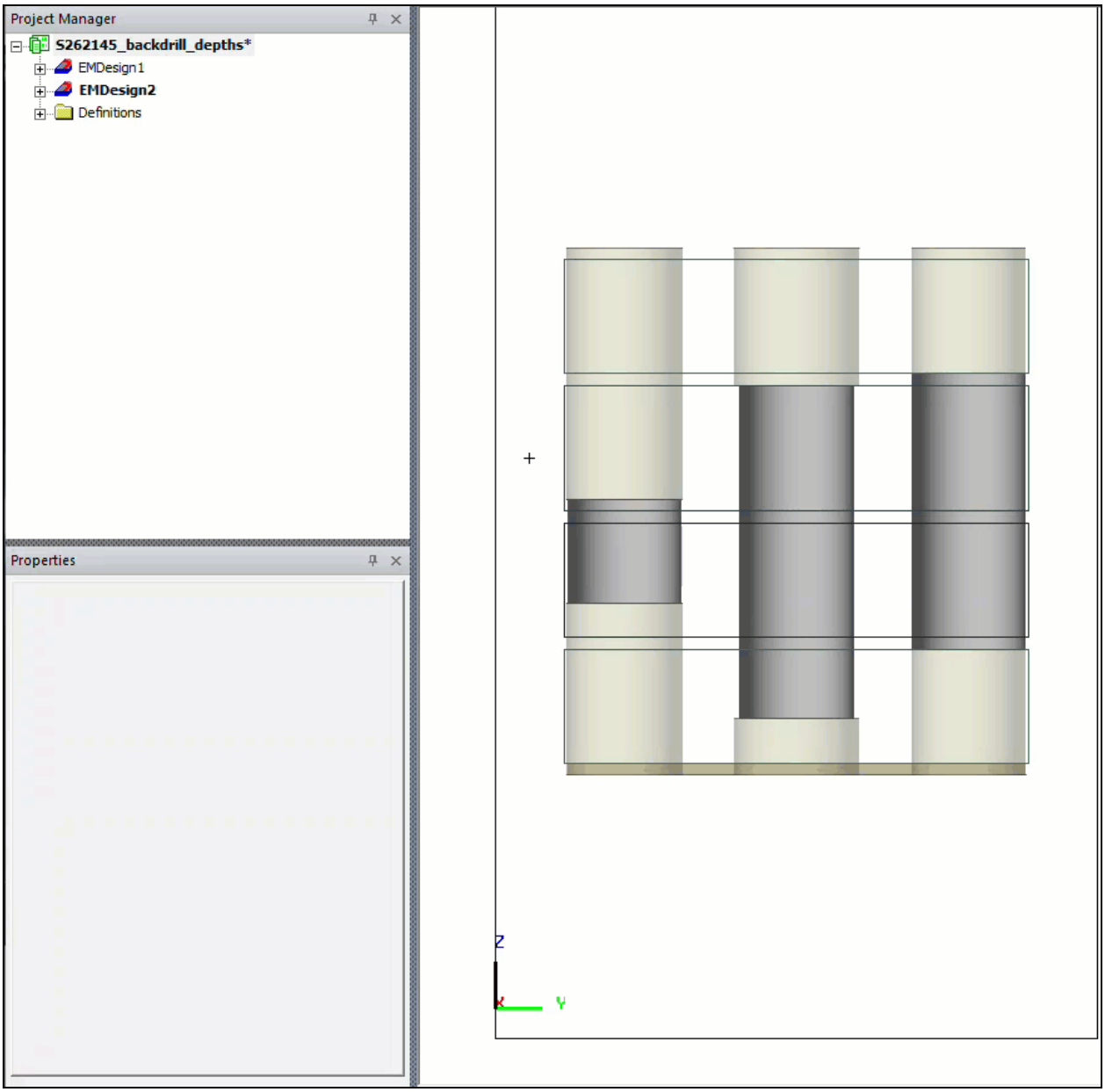
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. From the **Properties** window, the first row lists the padstack layers specified in the layer array variable. The second row designates the current index number (i.e., the designated padstack layer). The third row specifies the current offset (i.e., offset on the indexed padstack layer). Changing either of the numbers in the index and offset **Value** fields will

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dynamically change the model in the **Layout Editor**.



Using the Models Editor

1. To open the window for a model library, click **Tools** > **Edit Libraries** > **Models**. The **Edit Libraries** window opens on the **Models** tab.
2. From the **Models** tab, Click the **Libraries** scrolling window to select the library file for contents display. Use the buttons described to manage selected library objects:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Edit Model** – Edits properties or attributes for a selected model in the corresponding model editor. The resulting model may have the same name as the original, but it is saved to the current project, rather than back to the library. In effect, you have “checked out” the part to the project in order to edit it. If you want to write it back to the library, overwriting the original part, use **Export to Library**.

  (Click the **Show Project Definitions** check box to include current project elements in the **Edit Libraries** window display.)

  Some models – such as those that have been encoded – are not editable (button is unavailable). Encrypted models can be edited after the password has been entered to unlock it.

- **Add Model** – Creates a new model object in the selected library. The **Add Model** window allows you to name the model, and choose the model type you want to add. By default, the model is added via the appropriate model editor.
- **Clone Model(s)** – Creates a copy of selected object(s) with a different name in the selected library. Encoded models cannot be cloned (button is unavailable).
- **Remove Model(s)** – Removes selected object(s) on the library. System library models cannot be removed (button is unavailable).
- **Export to Library** – Exports a selected object to a different library. (Use also to export edited objects from the project to the library if necessary.) System library models cannot be exported (button is unavailable).

3. When finished, click **OK** to close the **Edit Libraries** window box.

Using the Material Editor

The Electronics Desktop **Material Editor** supports the viewing, modification, and saving of material information for substrate elements and stackup primitives. After modifying a material, the information can then be saved, used to define a design, or exported.

A material is a set of information that defines the physical properties of a substance — such as that used as a substrate, metallization, or solder mask layer — for its inclusion in a *stackup*, the physical foundation of a manufacturable model. Material definitions are stored in library files with an extension of **.amat**. You can access material definitions using the **Edit Material** window, also referred to as the **Material Editor**.

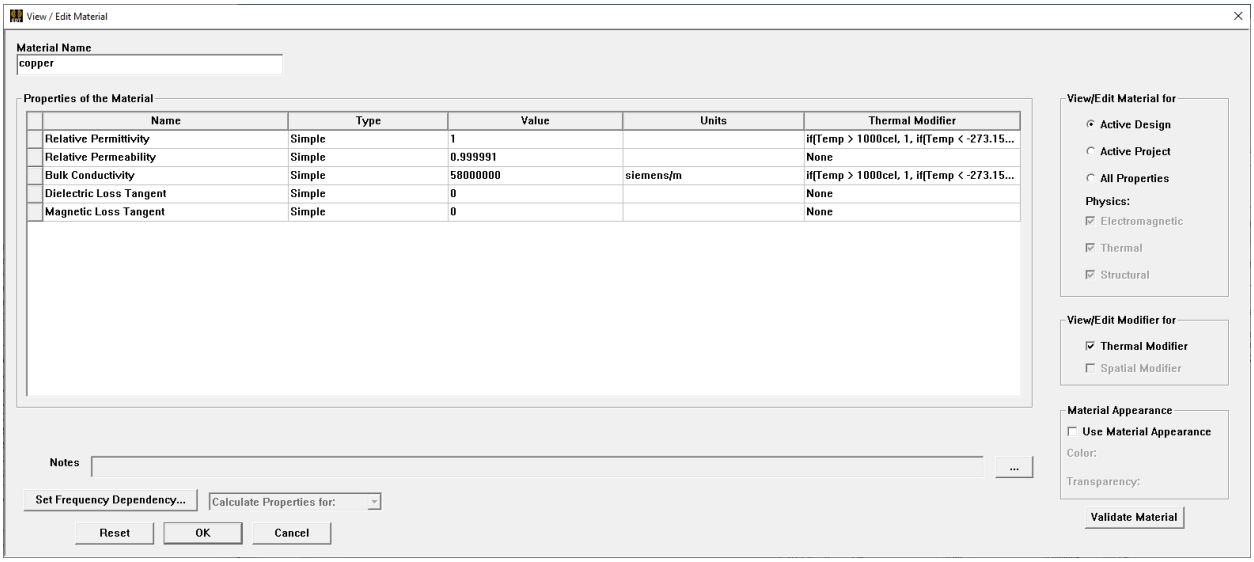
To start the **Material Editor**, select **Tools** > **Edit Libraries** > **Materials**. When the Edit Libraries window opens, use its controls to open the **Material Editor**.

The View / Edit Material Window

From the **View / Edit Material** window, control the property values of materials. Open the **View/Edit Material** window in one of 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.

- From **Tools**, select **Edit Libraries** > **Materials...** to open the **Edit Libraries** window. Then search for and double-click the name of the appropriate material, or highlight the name and select **View/Edit Materials**.
- From the **Project Manager** window, expand the project tree > **Definitions** > **Materials** folder. Then right-click the appropriate material and select **Edit Material...**.



Material Name — This box sets the material name. To specify or change the name, click in the box, then type the appropriate name.

Property Headings

Name — Displays the property name. The values in this column are not editable.

Type – Displays and sets the property type (**Simple** [the default] or **Anisotropic**). To change a properties Type setting, click in the **Type** cell and select the appropriate type.

Value — Displays and sets the property value. To change a property value, click the value you want to change, then type a value or parameter name.

Units – Displays and, where applicable, sets the unit that applies to the Value entry. (For example, the unit of magnetic saturation can be set to **Gauss**, **uGauss**, **Tesla**, or **uTesla**.) To change a unit, click it, then select the appropriate unit.

Property Names

Relative Permittivity – This box sets the material relative permittivity. To specify or change the value, click in the box, then type the appropriate value.

Relative Permeability – This box sets the material relative permeability. To specify or change the value, click in the box, then type the appropriate value.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bulk Conductivity – This box sets the material bulk conductivity. To specify or change the value, click in the box, then type the appropriate value.

Dielectric Loss Tangent — This box sets the material dielectric loss tangent. To specify or change the value, click in the box, then type the appropriate value.

Magnetic Loss Tangen t — This box sets the material magnetic loss tangent. To specify or change the value, click in the box, then type the appropriate value.

Magnetic Saturation – This box sets the material magnetic saturation. To specify or change the value, click in the box, then type the appropriate value.

Lande G Factor – This box sets the material Lande G factor. To specify or change the value, click in the box, then type the appropriate value.

Delta H – This box sets the material delta H. To specify or change the value, click in the box, then type the appropriate value.

View/Edit Material For

The check boxes in this group select the Ansys product(s) for which material properties are shown.

- Active Design
- This Product
- All Products

View/Edit Modifier For

The check boxes in this group select the Ansys product(s) for which modifier properties are shown.

- Thermal Modifier

Validate Material

Click **Validate Material** to validate the current material property values for **Electronics Desktop** product(s) checked. If validation succeeds, a green check mark appears under **Validate Now**. If validation fails, a red X appears instead, and an error message informs you of which parameter value(s) are invalid for which product(s), and why.

Set Frequency Dependency

Click **Set Frequency Dependency** to open a window that allows you to set options for the following:

- Piecewise Linear Input
- Loss Model Input

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Low-Loss Dielectric Model Input
- Enter Frequency Dependent Data Points

After you set options for any of the above, the **Value** column of the **MaterialEditor** indicates that a material's value is set to be frequency dependent, rather than set to a constant.

> **Note:** The **Set Frequency Dependency** option is available only with HFSS 3D Layout. To add a material with frequency dependence, or to edit the frequency dependence of a pre-existing material, you must first ensure that the active design selected in the **Project Tree** is an HFSS 3D Layout, otherwise **Set Frequency Dependency** is not available. For more information see Defining Frequency-Dependent Material Properties.

Material Editor window Controls

- Click **Reset** to restore all changed properties to their values prior to opening the **Material Editor**.
- Click **Cancel** to close the window without committing changes and return to the **MaterialBrowser**.
- Click **OK** to save changes and return to the **MaterialBrowser**.

For more information see Viewing and Modifying Material Attributes.

> **Note:** If you like the changes you have made in a material to be available for use in other projects, you must export the material to a library as described in the Edit Libraries window topic.

Editing an Existing Material

To edit an existing material, display its properties in the Edit Material window by doing either of the following:

- Open the **Material Browser** by clicking **Tools > Edit Libraries > Materials**, then search for and select the material you want to edit, and either double-click its name or click **View/Edit Materials**.
- In the **Definitions** /**Materials** subfolder of the **Project Tree**, locate the icon for the material you want to edit, then either double-click the icon or right-click it and select **EditMaterial**.

The Edit Material window opens (also called the **Material Editor**).

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 New Material

To create a new material, first open the **Material Browser** by clicking **Tools > Edit Libraries > Materials**, then do either of the following:

- Click **Add Material**. The Edit Material window opens (also called the **Material Editor**).
- Locate and select an existing material definition on which to base the new definition, then click **Clone Material(s)**. The selected definition is copied under a new name. To edit the new material, double-click its name to open the Edit Material window (also called the **Material Editor**).

Using Vendor Components

Electronics Desktop allows you to download vendor components from a server. After being downloaded, libraries are automatically configured so these components can be used in Electronics Desktop.

Some installations of Electronics Desktop include a set of RF Vendor Library components. To use these elements, you unzip the files into the **syslib/Vendor Elements** directory in **Electronics Desktop** installation.

Using RF Vendor Library Components

The RF Vendor library contains vendor-specific components for Electronics Desktop RF installations. The library is delivered as a compressed file, **Vendor_Libraries.zip**, in the **syslib/Vendor Elements** folder under your Electronics Desktop installation directory.

Here are the steps to follow to enable and place these elements.

Unzip the library directory:

1. Browse to the **syslib** folder in your Electronics Desktop RF installation. The directory path should be something like the following:

   `C:\Program Files\Ansys Inc\v`*`xxx`*`\AnsysEM\syslib`

   The entry, *xxx*, is the Ansys EM release number. The operating system field (*Win64* in the example, for the Windows OS) is *Linux64* for the Linux OS.

2. Ensure you have write permission in the **syslib** folder.
3. Open the folder **Vendor Elements**.
4. Locate the file **Vendor_Libraries.zip**.
5. Using any compression tool, extract the contents of the zip file to the **Vendor Elements** folder.

   The folder **Vendor_Libraries** should appear in the **Vendor Elements** 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:**
>
> Since configuration happens automatically at Ansys Electronics Desktop startup, if you unzip the archive while **Electronics Desktop** is open, you may need to save work, close, and restart **Electronics Desktop**.

Place the Component

1. In the **Component Manager** group box of Electronics Desktop, you should see a **Vendor Elements** directory with the **Vendor_Libraries** folder. Expand the folder, the **vendors** folder, to display a listing of the component types in the library (Amplifier, Capacitor, etc). Each component type represents a number of components from different vendors.
2. Double-click the icon for the type of part you want to place, then drag the symbol into the design area.
3. Right-click the component, and edit the **Properties**.
4. Select a **Vendor** on the listing of available vendors for the component type.
5. Select a **Series** on the listing available series on the selected **Vendor**.
6. Verify that the **Part** property has the part you wish to use. If the **Part** menu is not populated, or if the part you want is not in the list, change the **Data Type** property (**S-parameter**, **State-Space**, or **subcircuit**) until the part appears on the Part property menu.
7. Select a **Part** on the listing of available parts in the selected **Series**.
8. Optionally, use the **Sweep** window to select a text array variable containing part names to sweep while simulating the component. The text array variable must have been defined previously. See: Adding a Project Variable Array.
9. Click **OK** to close the **Properties** window.
10. Connect the component to the circuit.

Optionally, browse through the folders under **Vendor_Libraries** in the **syslib/Vendor Elements** directory. You can see what Vendors, Series, and Parts are available for each component type. The file extensions on the Part files indicate the **Data Type**:

- S-Parameter data types use the **.sNp** file extension.
- State-Space data types use the **.sss** file extension.
- Subcircuit data types use either the **.ckt** file extension or the **.lib** file extension.

Encrypted Libraries

Electronics Desktop lets users create personal, encrypted files of design information (typically in SPICE format) that they may be added to other designs via *.lib*, *.va*, or *.inc* statements. Encrypted library files are password-protected. If the user references an imported encrypted *.lib* or *.va* file with a user-defined password, the first time a component from the file 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.

Electronics Desktop prompts the user for the encryption password. The library file and password are then added to the list of password definitions in the **Tools** > **Password Manager** window.

> **Note:** For security reasons, encrypted components are not saved in the **Most Recently Used** list or the **Favorites** list.

Creating and Managing Encrypted Libraries

Electronics Desktop lets users create encrypted password-protected files (typically in SPICE format) that may be added to other designs via *.lib*, *.va*, or *.inc* statements. Access the encryption tools through the **Tools** > **Password Manager** window.

Password Assignment Types

From the **Enter Passwords** window, accessible on the **Password Manager** > **New...** option, an encrypted file may be assigned a choice of password types:

- **Enter password**: User-defined passwords for an encrypted library file. Entering and confirming **Full Access** passwords grants users simulation, read, and edit permissions, while **Execute Only Access** permits users to add components and simulate a design, but they cannot read or edit the contents.
- **Use Ansys Password (Execute Only)**: A generic, hard-coded execute-only password shared across all Ansys products. Any Ansys product may read an applicable file encrypted in any other Ansys tool using the **Ansys Password**, making it the easiest way to share encrypted library files among Ansys applications if the files are not expected to be viewed or editable.
- **Use Same Password as**: Choose an existing resource on the drop-down menu to use the same password settings and synchronize passwords across multiple library 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:

1. Passwords are not saved between sessions and must be re-entered each time **Electronics Desktop** is started. Passwords *are* available after one project is closed and another is opened, if the application is not restarted.
2. **Electronics Desktop** and Nexxim use the Advanced Encryption Standard (AES) to encrypt libraries and models. **Electronics Desktop** and Nexxim use a 128 bit block cipher with a 256 bit key.
3. You can encrypt selected parts of a file by enclosing them within a protected "block". You can set up multiple .protect/.unprotect blocks in the same file, but you cannot nest them.

   e.g.,

   .protect

   This block is encrypted.

   .unprotect

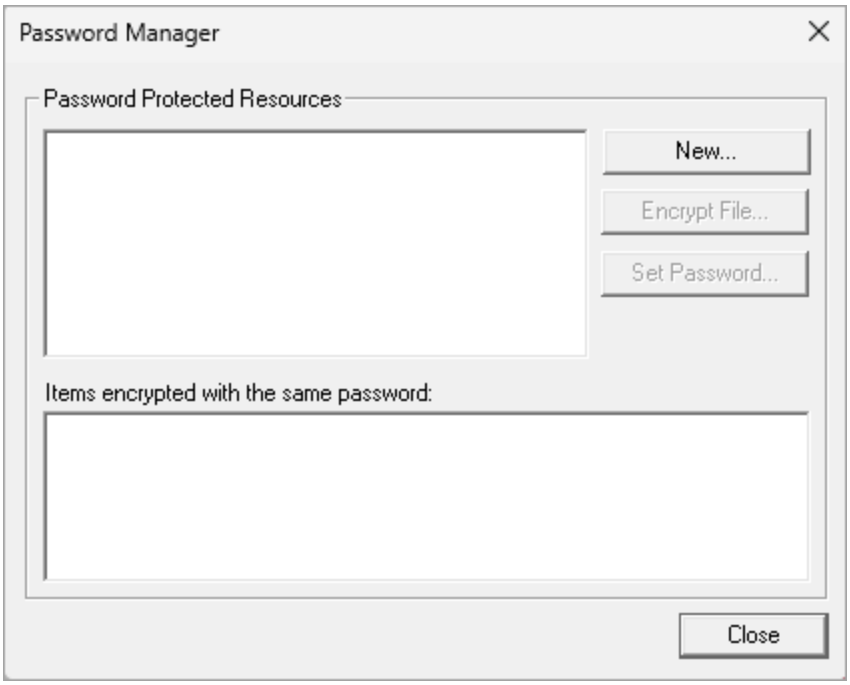
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up Passwords and Encrypted Libraries

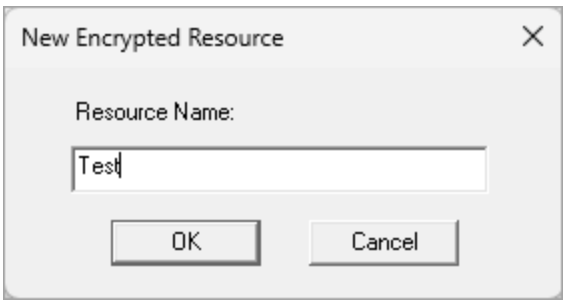
Because an encrypted library requires a password, encryption is done through the **Password Manager** window. The **Password Manager** windows lets users create a password or password set as a "password protected resource", which is a group of encrypted files which use the same password. Once the resource and its password type is defined, select it as the password for one or more newly encrypted libraries.

Generate a New Password Protected Resource

1. From **Tools** menu, select **Password Manager...** to open the **Password Manager** window.



2. Click **New...** to open the **New Encrypted Resource** window.

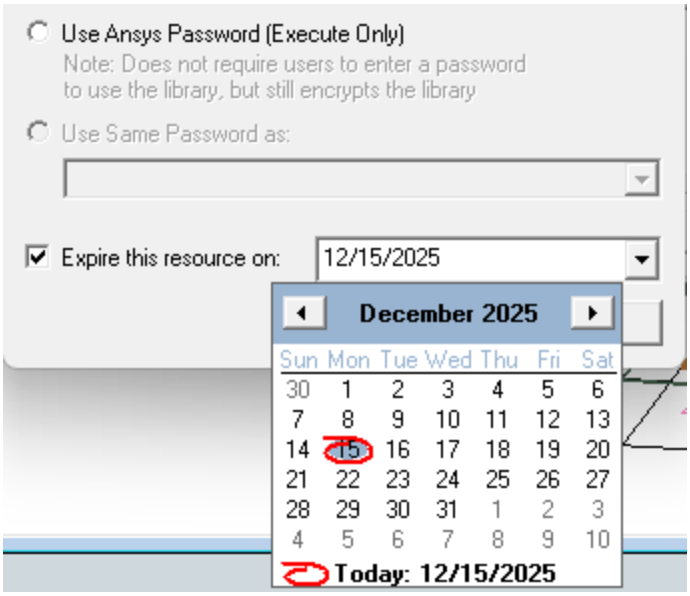


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

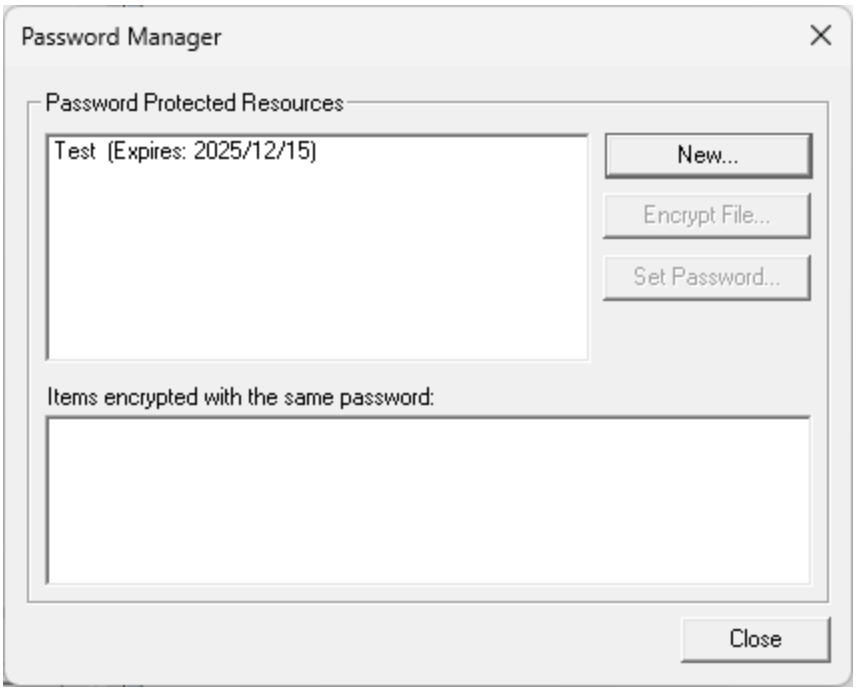
3. Type a name for the resource into the field. Then click **OK** to simultaneously close the **New Encrypted Resource** window and open the **Enter Passwords** window.

4. Choose on the following:
    - **Enter password**: choose between **Full Access** and **Execute Only Access**, then enter and confirm the new password in the given text fields.
    - **Use Ansys Password (Execute Only)**: as stated, this choice does not require users to enter a password to use the library, but still encrypts the library.
5. Optionally, assign an expiration date for the encrypted library by checking the **Expire this resource on** box and selecting a date on the drop-down calendar. If an expiration date is assigned to a library during encryption, the library cannot be used after the assigned date.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

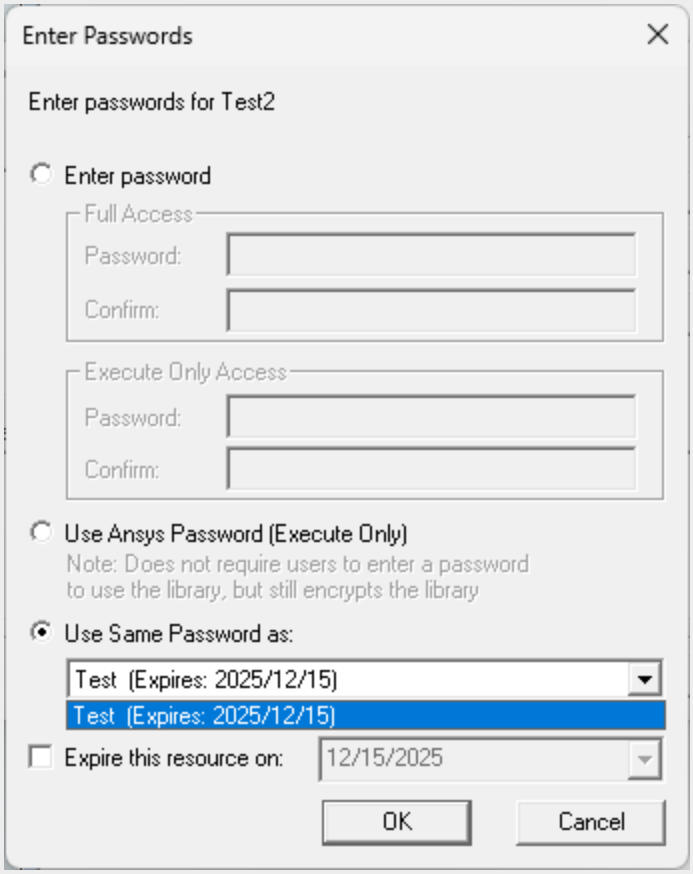


6. Click **OK** to add the new resource to the **Password Manager** list and simultaneously close the **Enter Passwords** window. Any number of libraries may be assigned to the new resource.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Once one or more resources have been added to the **Password Protected Resources** list on the **Password Manager** window, **New** resources can use the same password profile. From the **Enter Passwords** window, select **Use Same Password as**, which was previously unavailable, and choose the existing resource on the drop-down menu.

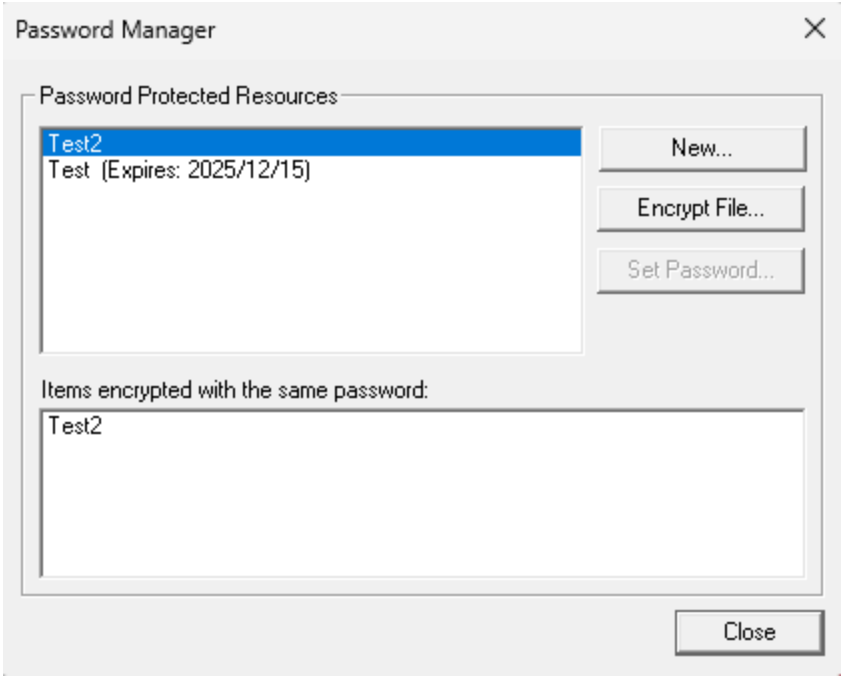


Generate a New Encrypted Library

When a library file is encrypted, the user can choose to overwrite the existing target library with the encrypted one, or save the encrypted library file with a new name and/or to a new location. During encryption, the user is prompted to choose whether the process respects .prot[ect] statements or ignores them. To generate a new encrypted library, complete the following steps.

1. From the **Password Manager** window, select a resource on the **Password Protected Resources** list to enable the **Encrypt File...** button.

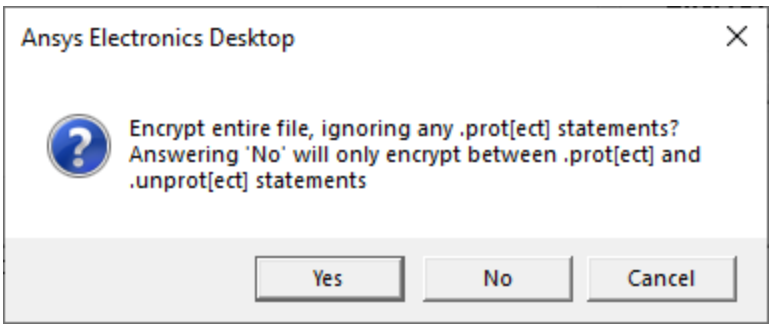
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Encrypt File...** to open an explorer window. Navigate to the appropriate library file.

> **Note:** Only library files with **Circuit (*.*lib*)** and **Verilog (*.va*)** file extensions are valid.

3. Select a library file, or first narrow the available choices by selecting **Circuit Files (*.*lib*)** or **Verilog Files (*.va*)** on the **Files of type** drop-down menu. Then click **Open** to immediately open the prompt: **Encrypt entire file, ignoring any .prot[ect] statements?**



4. As stated in the prompt, choose on the following:
   - Click **Yes** to permanently encrypt the entire library file.
   - Click **No** to recognize any **.prot[ect]** statements and encrypt only the remaining

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

aspect.

- Click **Cancel** to exit out of the prompt and return to the previous explorer window.

5. After the library file is encrypted, an explorer window will appear. Navigate to an appropriate location to save and/or rename the file, if appropriate. If the encrypted library file is saved to the same location and with the same file name as the original unencrypted library file, the unecrypted file is overwritten permanently.
6. Click **Save** to close the explorer window and create the encrypted library file.

Using the Library Editor

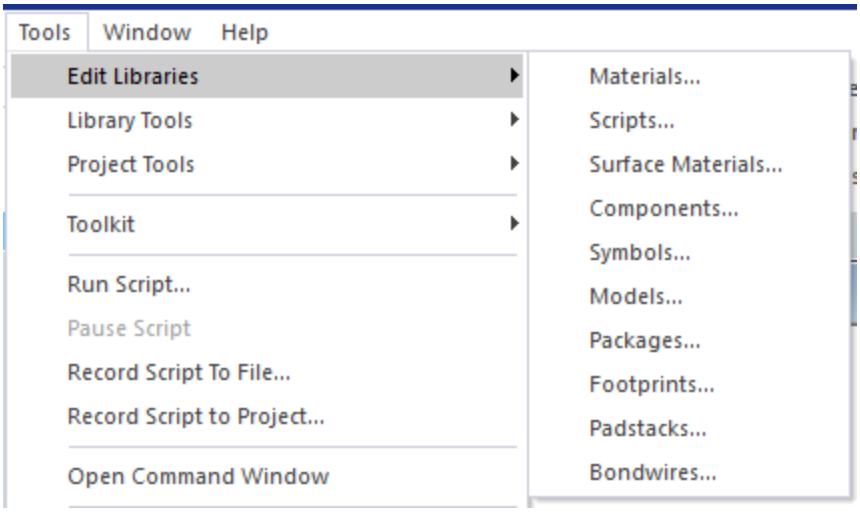
The Electronics Desktop **Library Editor**, also called the **Edit Libraries** window, allows you to access, manage, save, and control the contents of various development libraries which contain information and properties for the following design entities:

- Materials
- Surface Materials
- Scripts
- Components
- Symbols
- Models
- Packages
- Footprints
- Padstacks
- Bondwires

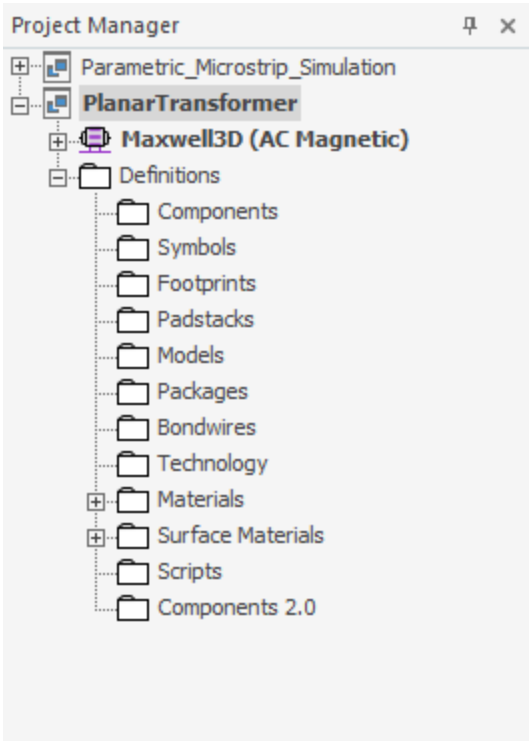
Starting the Library Editor

To start the **Library Editor**, select **Tools** > **Edit Libraries**, then select the type of library you wish to access on the following menu:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



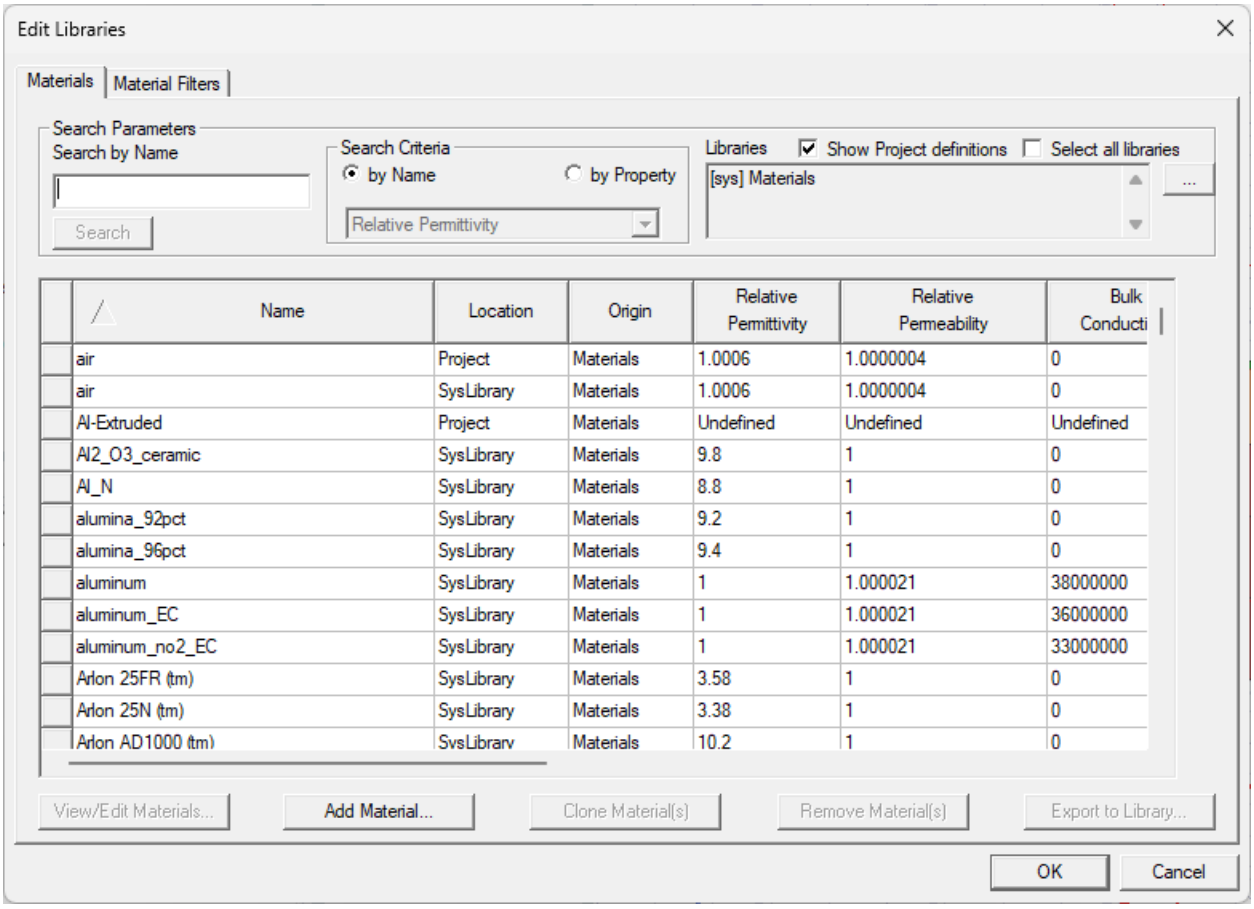
You may also right-click any of the folders listed in the **Definitions** directory of the **Project Tree** in the **Project Manager** window, then select **Edit Library**.



Edit Libraries Window

After choosing a library to edit, the **Edit Libraries** window opens with a tab open to the library you selected, such as the **Materials Browser**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Edit Libraries** browser searches library objects using filters based on a number of attributes:

- Name
- Property
- Model type

You can customize the **Edit Libraries** window by clicking **Show Project definitions** and **Show all libraries** (to override filtering).

The following **Edit Libraries** browser controls are available:

- **Edit** – Edits properties/attributes for the selected object. The resulting object can have the same name as the original, but it is saved to the current project, rather than back to the library. In effect, you have “checked out” the part in order to edit it. If you want to write it back to the library and overwrite the original part, use **Export to Library**.
- **Add** – Creates a new object in the selected library.
- **Clone** – Creates a copy of a selected object with a different name in the selected library.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Remove** – Removes an object on the library
- **Export** – Exports edited objects from a project to a library, or exports selected objects from one library to another. If the component you are exporting is hierarchical, after you click to **Export** and specify a library name in the **Export to user library** window to open the **Export Hierarchy** window. For more information see Exporting Hierarchical Components.

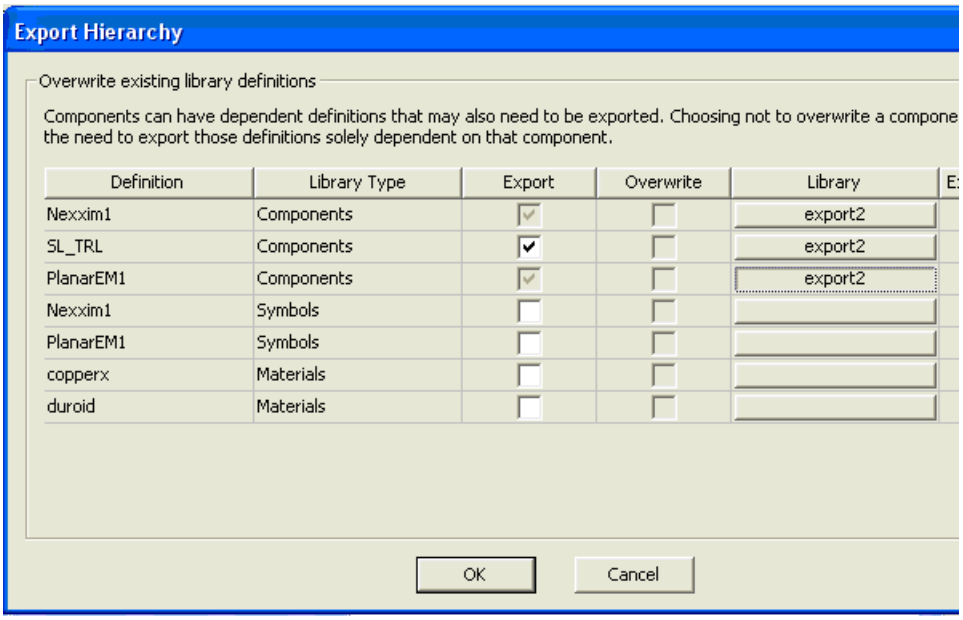
Note:

1. When you modify a library component, the modified definition is automatically transferred to the components available to the current project. In effect, you have "checked out" the component to modify it, and it is now in the project. The original library definition is intact. To update the library definition, you must do the extra step of exporting the modified component on the project to the library.
2. Subcircuits automatically inherit higher-level library configurations.
3. Project variables and project datasets used in designs represented by hierarchical components is saved to the library with the component and restored when the component is used. If the variable or dataset already exists in the project (determined by name), the library version is ignored. When creating project variables or datasets for hierarchical component designs to be saved in a library, ensure the names of the variables and datasets are appropriate and not likely to clash with those of other hierarchical components or common defaults.

Exporting Hierarchical Components

Electronics Desktop allows you to export hierarchical components, along with their solutions. Click **Export** to export a hierarchical component in the Library Editor and specify a library name in the **Export to user library** window to open the **Export Hierarchy** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The component selected to be exported appears in the top-most row of the window. The library to which it is exported is listed beneath the **Library** column (in this example, “export2”). To select a different library to export to, click the button in the **Library** column (in this example, “export2”).
- Dependent definitions appear underneath the top-most row of the window and may also be exported. Definitions that are mandatory for export has an unavailable **Export** check box.
- You can export a component’s solution by checking its **Export** group box. At import time, the solution is re-associated with the design.
- Symbols need not be exported, they is auto-generated at import time.
- Each definition is exported as a definition in its own right. That is to say, at import time, using the above example, "Planar EM1" is visible in the "export2" library as a component independent of "Nexxim1".
- Components that originated in libraries (e.g., the "SL_TRL" component) need not be exported. At import time, each library component is retrieved from its original library and re-associated with 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12 - 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 HFSS 3D Layout. 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 HFSS 3D Layout 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.

Note:

For **HFSS** and **HFSS 3D Layout** designs only, you can validate all design variations in a parametric setup prior to performing the parametric analysis. See: Validating 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.

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

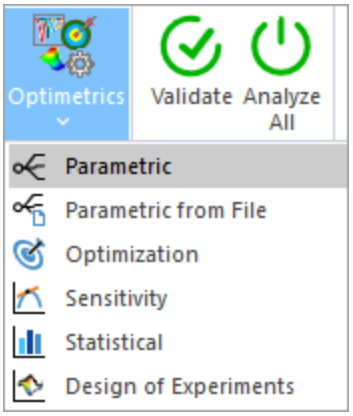
A *parametric setup* specifies all of the design variations that Optimetrics drives the software to solve. A parametric setup is made up of one or more variable sweep definitions, which are a set of variable values within a range that you want the software to solve when you run the parametric setup.

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 **HFSS 3D Layout** > **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

3. Optionally, validate the parametric design variations before solving the parametric analysis.

   The **Validate** command is located in the shortcut menu that appears when you right-click the parametric setup name. The results of the validation process can be viewed by selecting **View Analysis Result** from the same shortcut menu. During validation, messages appear in the Message Manager window, and the validation progress is shown in the Progress window.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

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 **HFSS 3D Layout** > **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.

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 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:

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.

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 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Do one of the following:
    - To modify a variable value, click a value text box in the table and type a new value.
    - To delete a variable value from the sweep definition, click the row you want to delete, and then click **Delete**.
    - To add a new variable value to the sweep definition, click **Add**. Then click in the value text box and type a new value.

    > **Warning:**
    >
    > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Your modifications are tracked and available for viewing at the bottom of the **Setup Sweep Analysis** dialog box under the **Sweep Definitions** tab. The operations you performed are listed with descriptions.

> **Warning:**
>
> If you modify an original sweep definition using the **Add/Edit Sweep** dialog box after you have manually modified its table of design variations, your manual modifications become invalid and are removed. A warning is displayed to inform you that your manual values are about to become invalid, so you can decide whether or not to proceed.

Overriding a Variable's Current Value in a Parametric Setup

If you choose not to sweep a variable, the variable's current value set for the nominal design is used when it solves the parametric setup. To override the current variable value for a parametric setup:

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.

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 select the **Override** option first, and then type a new variable value in the **Value** box.

4. Optionally, click a new unit in the **Units** box.

To revert to the current variable value, clear the **Override** option.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Specifying a Solution Setup for a Parametric Setup

To specify the solution setup that the software analyzes when it solves a parametric setup:

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.
2. Select the solution setup you want the software to use when it solves the parametric setup.

   The parametric setup is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

Specifying the Solution Quantity to Evaluate for Parametric Analysis

When you add a parametric setup, you can identify one or more solution quantities to be presented in the *Post Analysis Display* dialog box. The solution quantities are specified by mathematical expressions that are composed of basic quantities, such as output variables. When you view the results, HFSS 3D Layout 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.

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 **Add Calculation** to add the expression in the *Add/Edit Calculation* dialog box's **Calculation Expression** field to the Calculations tab of the *Setup Sweep Analysis* dialog box.
4. Click **Done** to close the *Add/Edit Calculation* dialog box.

Setting up Calculations for Optimetrics

The **Setup** dialog boxes for each of the Optimetrics types include a **Setup Calculations** button. Clicking this button displays the **Add/Edit Calculation** dialog box, which contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**.

The **Context** pane contains fields for the Report Type to use, the Solution, and depending on the Report Type selection, the Geometry.

The **Trace** tab contains fields for the Calculation expression and, to build the expression, a Category list, a Quantity list with a Text Filter field, and a list of Functions available for the selected Category. The Range function button opens a dialog box in which you can define a range function to apply a function to the expression.

The Category list for the **Trace** tab includes Variables and Output Variables. An Output Variables... button lets you open a dialog box to define and edit the Output Variables.

To set up an Optimetrics calculation:

1. Click **Setup Calculations** to open the **Add/Edit/Calculation** dialog box.
2. In the **Report Type** text field in the **Context** pane, select from the drop-down menu of available types.

   Selecting Fields as the Report type causes the **Geometry** field to display.
3. In the **Solution** text box, select from the drop-down menu of available solutions.
4. If the **Geometry** field is available, select from the drop-down menu.
5. In the **Trace** tab, specify the solution Category, a Quantity, and Functions. The resulting expression will be displayed in the *Calculation Expression* field.
   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.

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 create an output variable that represents the solution quantity, do the following:

i. Click **Output Variables**.

The **Output Variables** dialog box appears.

ii. Add the expression you want to evaluate, and then click **Done**.

The recently created output variable appears in the **Quantity** list.

iii. Click a new output variable in the **Quantity** drop-down menu.

> **Note:**
>
> The calculation you specify must be able to be evaluated into a single, real number.

The selected Quantity appears in the **Calculation Expression** field.

c. Select the **Function** from the list.

The selected function is applied to the **Quantity** in the **Calculation Expression** field.

6. To apply a **Range function** to the **Calculation Expression**, see Setting a Range function.
7. Click **Add Calculation** to add the expression in the **Add/Edit Calculation** dialog box's **Calculation Expression** field to the Calculations tab of the **Setup Sweep Analysis** dialog box.
8. Click **Done** to close the **Add/Edit Calculation** dialog box.

Specifying a Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the value of intrinsic variables such as frequency (F) at which the solution quantity will be extracted. For a parametric setup, the 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.

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 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.
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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

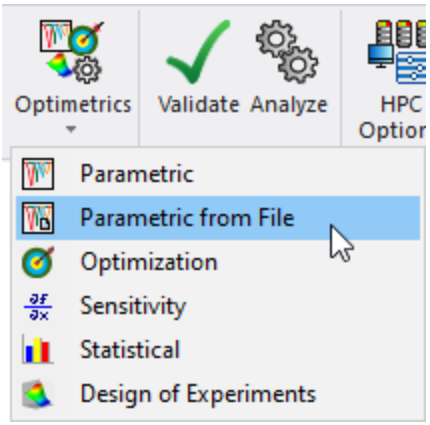
   - Right-click the progress window, and select **Show Subtask Progress Bars**.

     To hide the child progress bars:

   - 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 **HFSS 3D Layout** > **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:

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[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.

Validating a Parametric Setup

For HFSS and HFSS 3D Layout designs, you can validate all design variations in a parametric setup before solving the parametric analysis. This process is separate from the **HFSS 3D Layout**> **Validation Check** command or the equivalent **Validate** icon on the **Simulation** ribbon tab, which only validate the nominal solution setup.

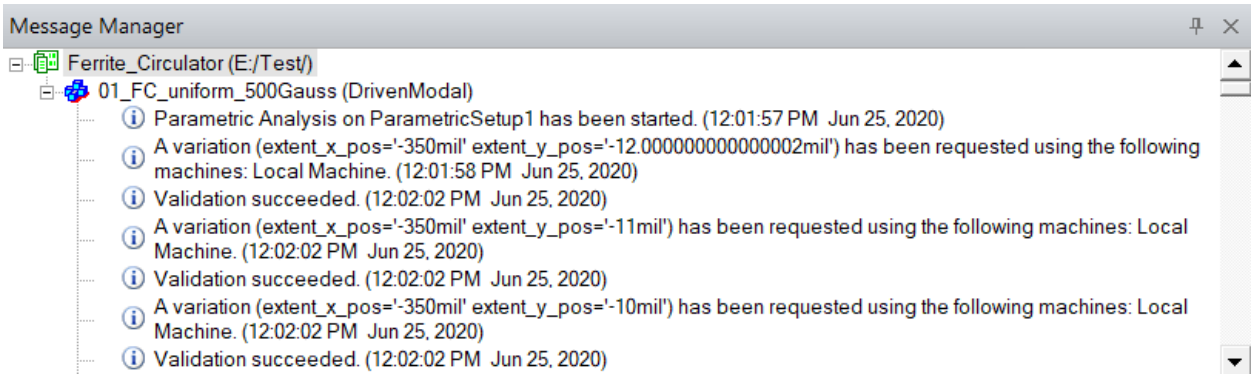
After creating a parametric analysis setup, validate the design variations using the following procedure:

1. Under *Optimetrics* in the Project Manager, right-click the ***parametric setup name*** and choose **Validate** from the shortcut menu.

   The design variations are validated one at a time until all have been checked.

2. Optionally, display the **Progress** window to see a visual indication of the validation progress.
3. Optionally, display the **Message Manager** window to see the validation messages for 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.



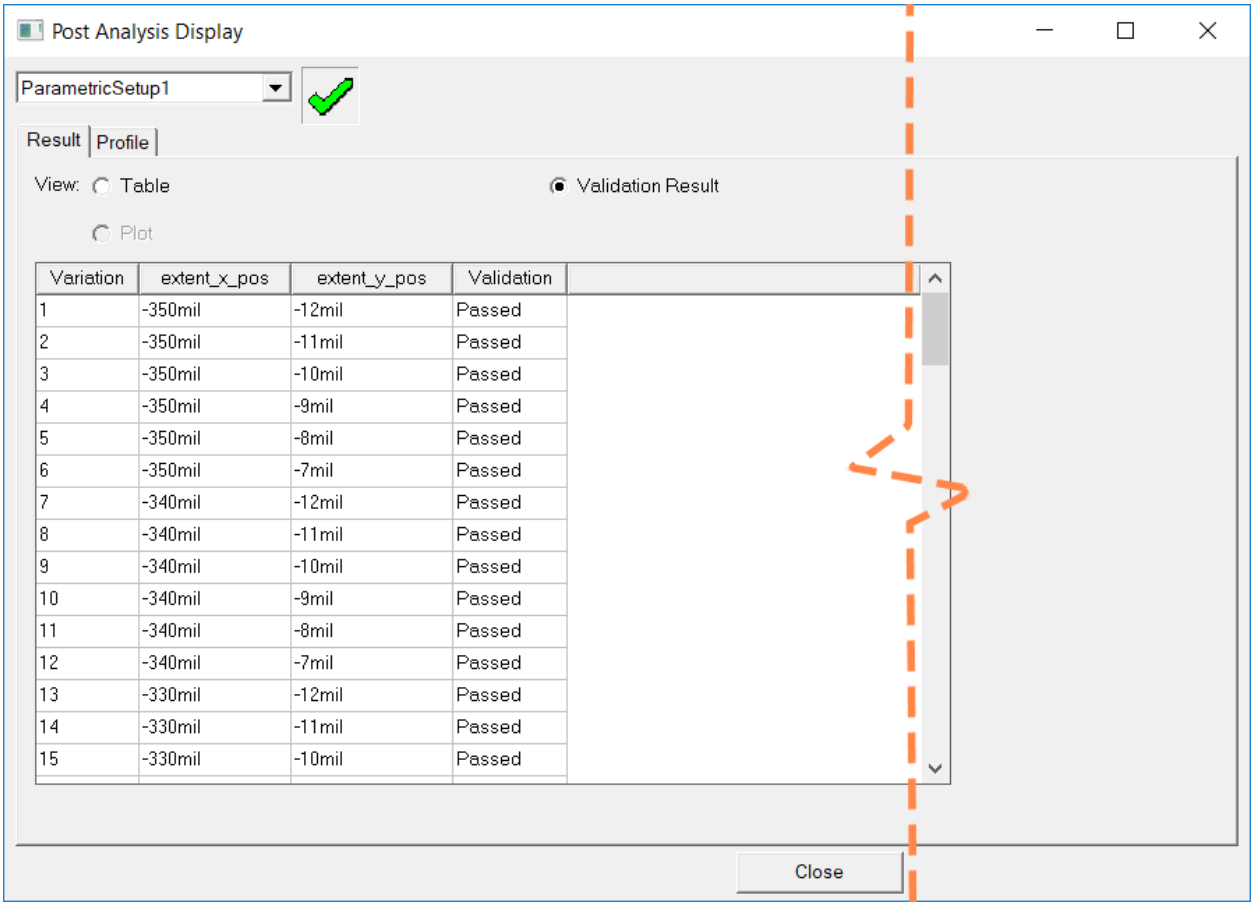
4. Under *Optimetrics* in the Project Manager, right-click the ***parametric setup name*** and choose **View Analysis Result**.

   The *Post Analysis Display* dialog box appears with the **Result** tab selected and with **Validation Result** selected as the *View* option.

5. If you have added more than one parametric setup to the design, select the ***setup name*** you wish to review from the drop-down menu in the upper-left corner of the dialog box.

   The following example shows a validation result where all variations passed:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



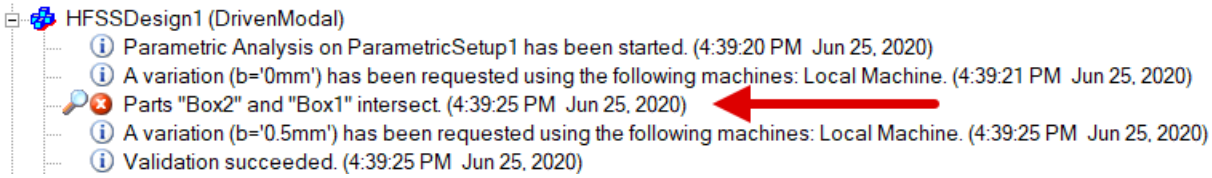
Variation	extent_x_pos	extent_y_pos	Validation
1	-350mil	-12mil	Passed
2	-350mil	-11mil	Passed
3	-350mil	-10mil	Passed
4	-350mil	-9mil	Passed
5	-350mil	-8mil	Passed
6	-350mil	-7mil	Passed
7	-340mil	-12mil	Passed
8	-340mil	-11mil	Passed
9	-340mil	-10mil	Passed
10	-340mil	-9mil	Passed
11	-340mil	-8mil	Passed
12	-340mil	-7mil	Passed
13	-330mil	-12mil	Passed
14	-330mil	-11mil	Passed
15	-330mil	-10mil	Passed

If a design variation fails the validation check, that row of the table is rendered with red text, and the *Validation* column shows "Failed":

Variation	b	Validation
1	0mm	Failed
2	0.5mm	Passed
3	1mm	Failed
4	-0.5mm	Passed

The details of the failure are not given in the *Post Analysis Display* dialog box. Look in the **Message Manager** window for those details:

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 preceding example, two of the four design variations specified a dimensional variable (*b*) that caused two parts to intersect. That is, geometric interference was detected for two design variations.

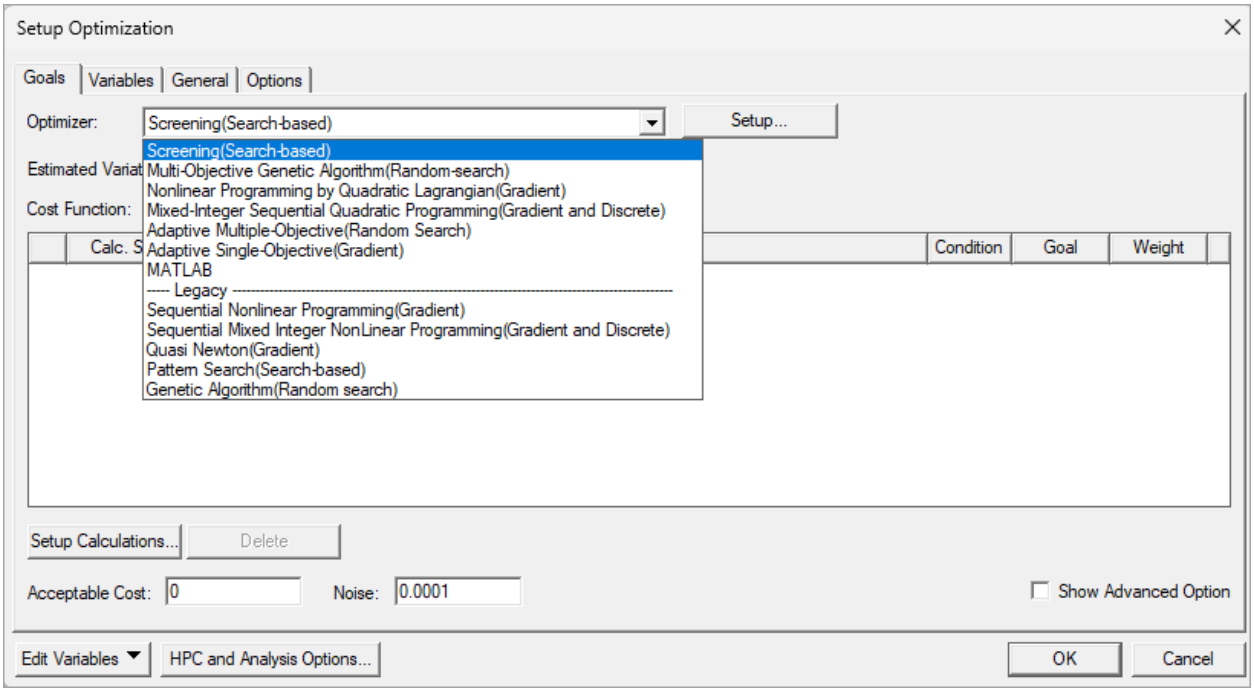
6. Click **Close** to dismiss the *Post Analysis Display* dialog box.

Optimization Overview

Optimetrics interfaces with Ansys Electromagnetics products to help you optimize a wide variety of design parameters based on variable geometry, materials, excitations, component values, etc. Optimization is the process of locating the minimum of a user-defined cost function. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

Choosing an Optimizer

Conducting an optimization analysis allows you to determine an optimum solution for your problem. In optimization analyses, there are many available optimizers.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Goal-Driven Optimizers

These use a Decision Support Process (DSP) based on satisfying criteria as applied to the parameter attributes using a weighted aggregate method. In effect, the DSP can be viewed as a post-processing action on the Pareto fronts as generated from the results of the various optimization methods.

- Screening (Search based) – a non-iterative direct sampling method that uses a quasi-random number generator based on the Hammersley algorithm. You can start with Screening to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value. Usually Screening is used for preliminary design, which can lead you to apply one of the other approaches for more refined optimization results.
- Multi-Objective Genetic Algorithm – an iterative random search algorithm that can optimize problems with continuous input parameters. It is better for calculating the global optima. You can start with MOGA to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value.
- Nonlinear Programming by Quadratic Lagrangian (Gradient) – a gradient-based, single-objective optimizer based on quasi-Newton methods. Ideally suited for local optimization.
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) – a gradient-based, single-objective optimizer that solves mixed-integer non-linear programming problems by a modified sequential quadratic programming (SQP) method. Ideally suited for local optimization.
- Adaptive Multiple Objective – an iterative, multi-objective optimizer that employs a Kriging response surface and MOGA. In this method, the use of a Kriging response surface allows for a more rapid optimization process because all design points are not evaluated except when necessary and part of the population is simulated by evaluations of the Kriging 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SNLP and SMINLP optimizers, the relationship can be linear or nonlinear. For the Quasi Newton and Pattern Search optimizers, the relationship must be linear.

Cost functions can be quite nonlinear. As a result, during the function evaluations of the algorithm, the cost function can vary significantly. Also, it is important to understand the relationship between optimization function evaluation and iteration. Every iteration, depending on the number of parameters to be optimized, performs several function evaluations. These function evaluations, depending on how nonlinear the cost function is, could show drastic changes. The presence of drastic changes has no bearing on whether the optimization algorithm converged or not.

In the case of non-gradient search-based optimization algorithms, such as "pattern search," which are entirely based on function evaluations, one could see drastic changes in the function evaluations depending on how nonlinear the cost function is. This could seem misleading as if the algorithm did not converge since in theory one expects the cost function to decrease from one iteration to the next. The optimetrics, however, reports function evaluations and not necessarily the optimizer performance per iteration.

> **Note:**
>
> The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.

Legacy Optimizers

These include

- 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.

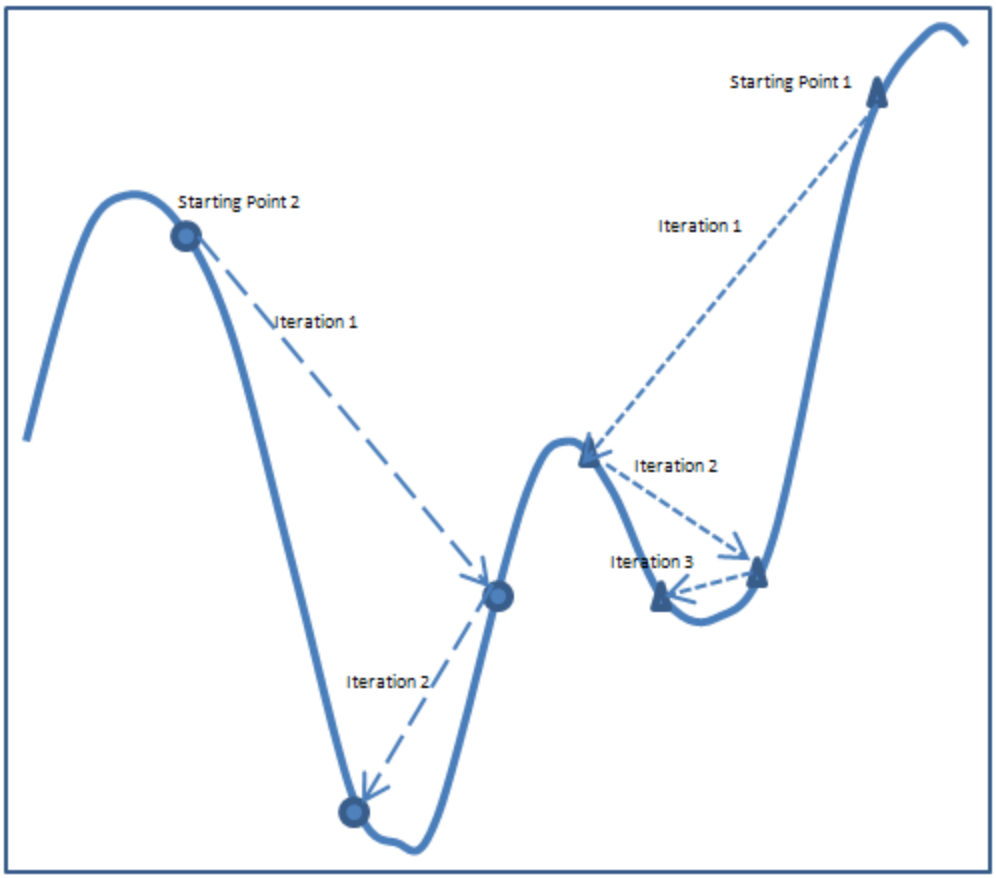
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sir Isaac Newton first showed that the maximum or minimum of any function can be determined by setting the derivative of a function with respect to a variable (x) to zero and solving for the variable. This approach leads to the exact solution for quadratic functions. However, for higher order functions or numerical analysis, an iterative approach is commonly taken. The function is approximated locally by a quadratic and the approximation is solved for the value of x. This value 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.

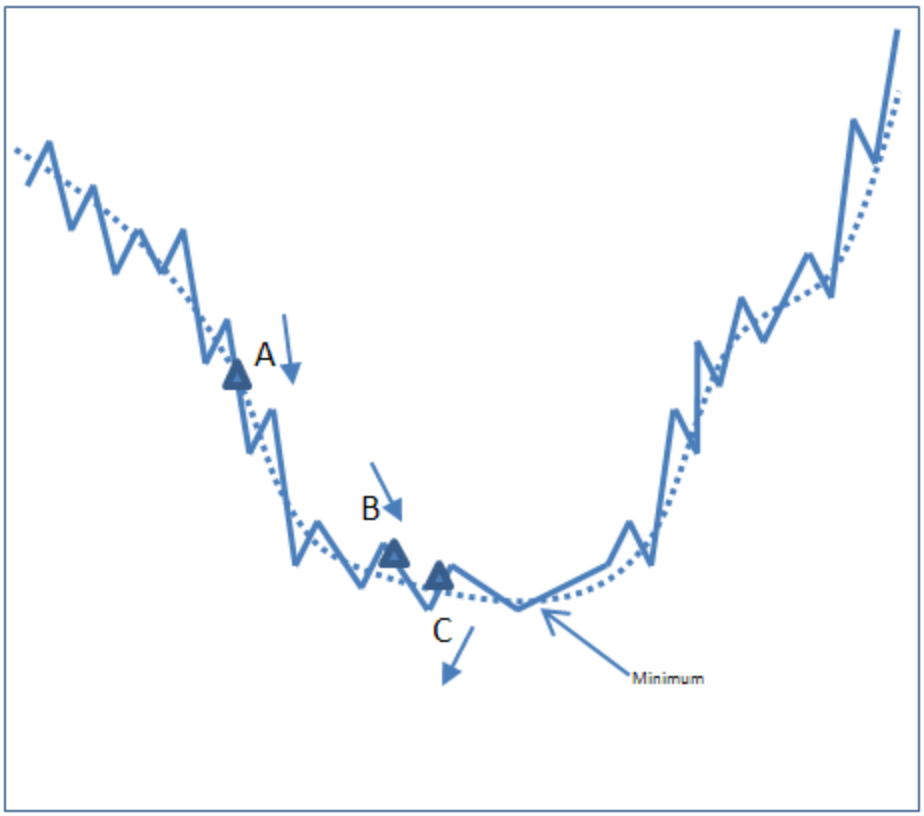
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

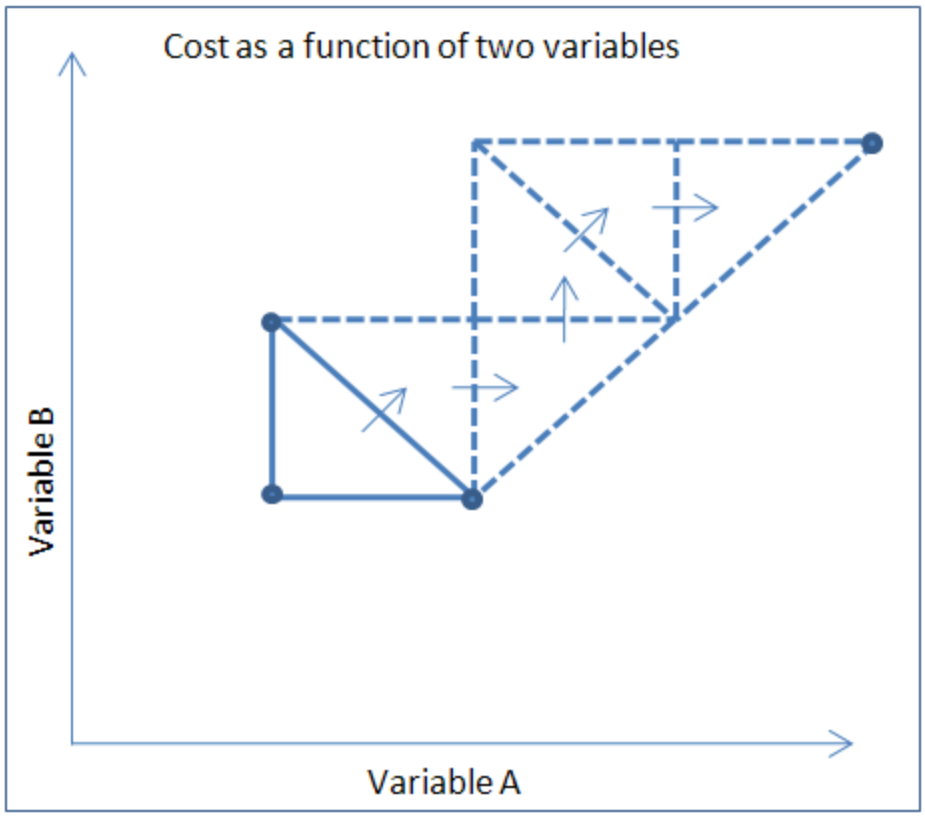
The cost value is calculated at the vertices of the simplex. The optimizer mirrors the simplex across one of its faces based on mathematical guidelines and determines if the new simplex provides better results. If it does not produce a better result, the next face is used for mirroring and the pattern 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.

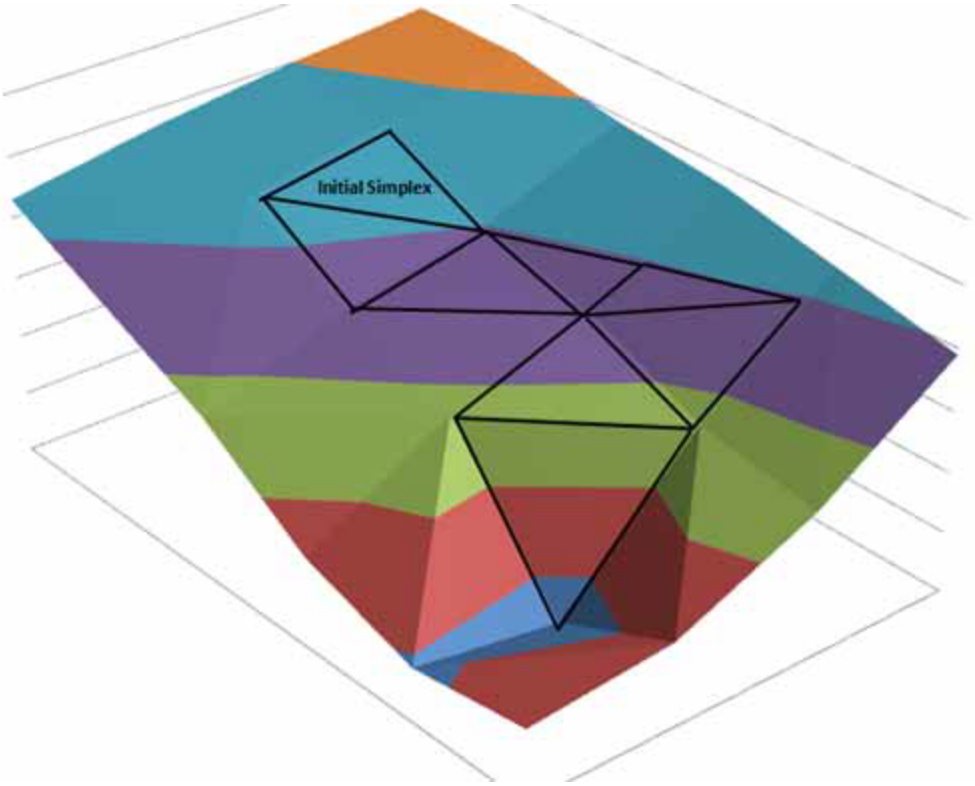
> **Note:**
>
> The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.

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.

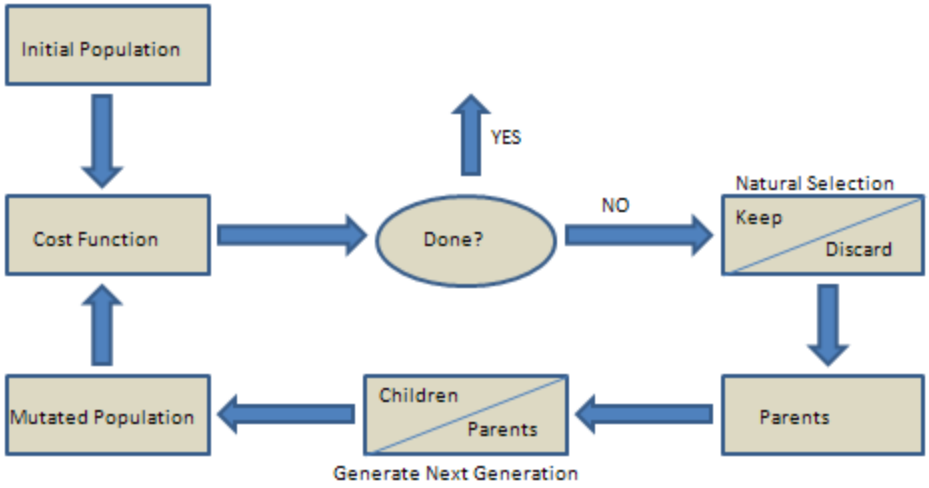
resembles the setup for SNLP, except that you must flag the supporting integer variables. You can set up internal variables based on the integer optimization variable.

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
```

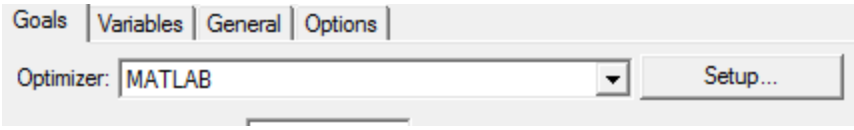
The answer will be 1 if the MATLAB Optimization Toolbox is licensed or 0 otherwise.

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.

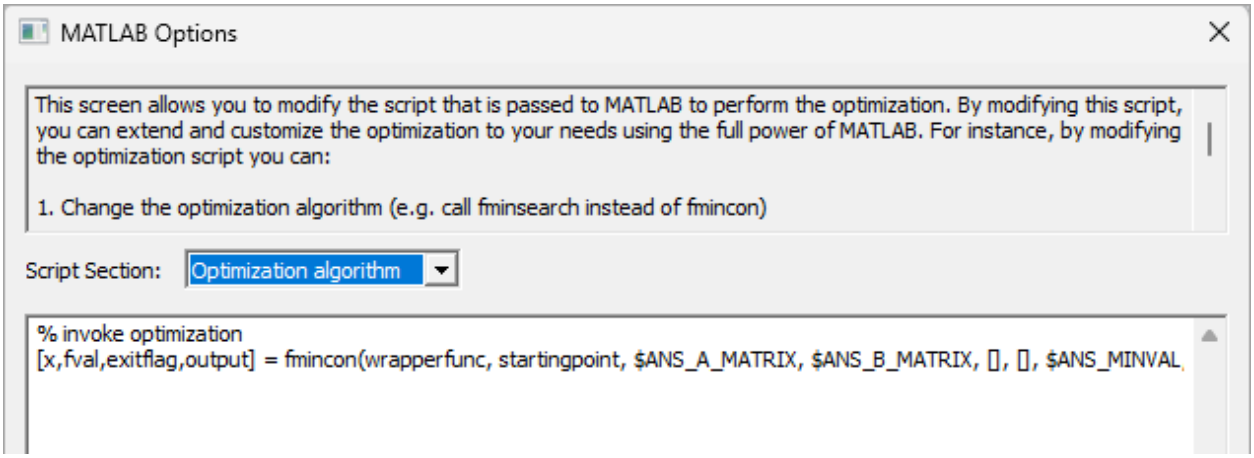
MATLAB Optimization Setup

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.

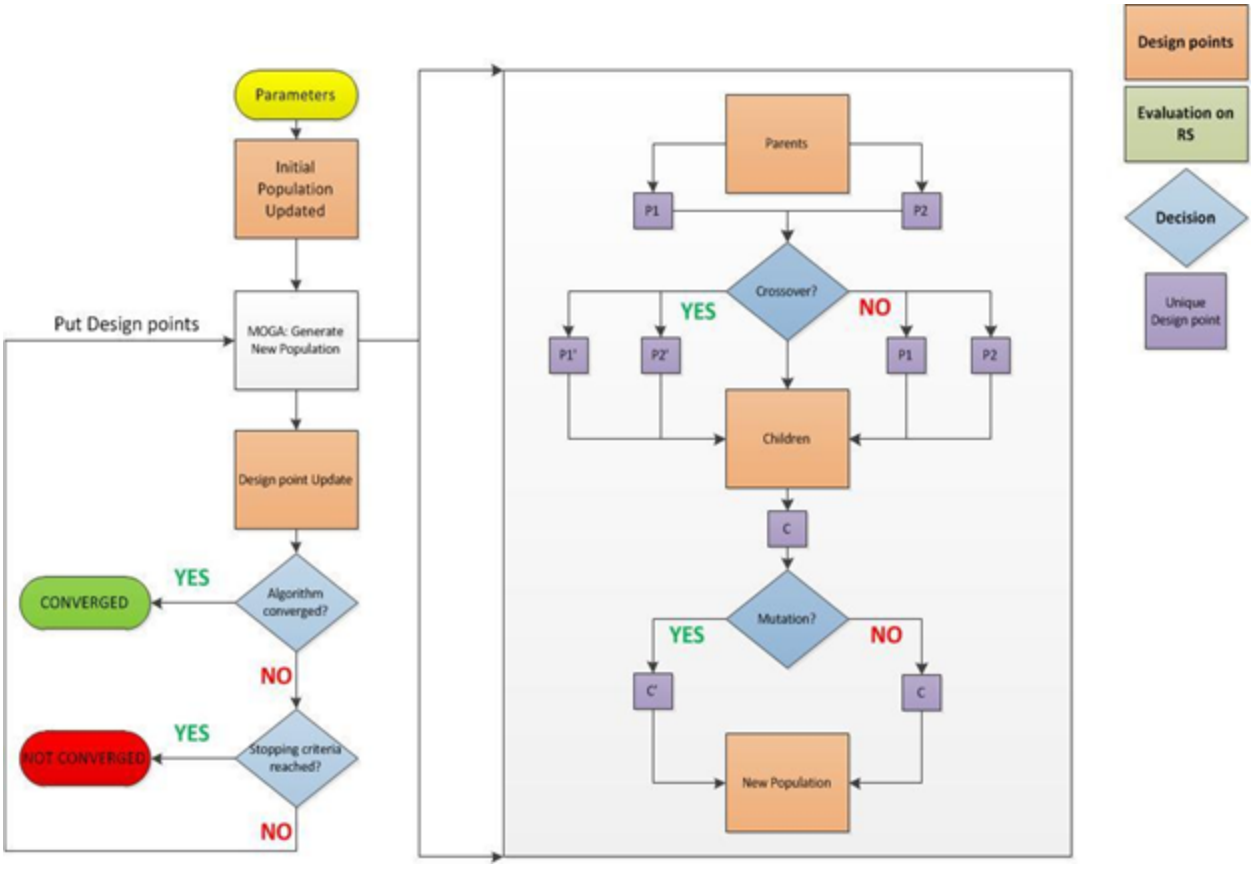
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

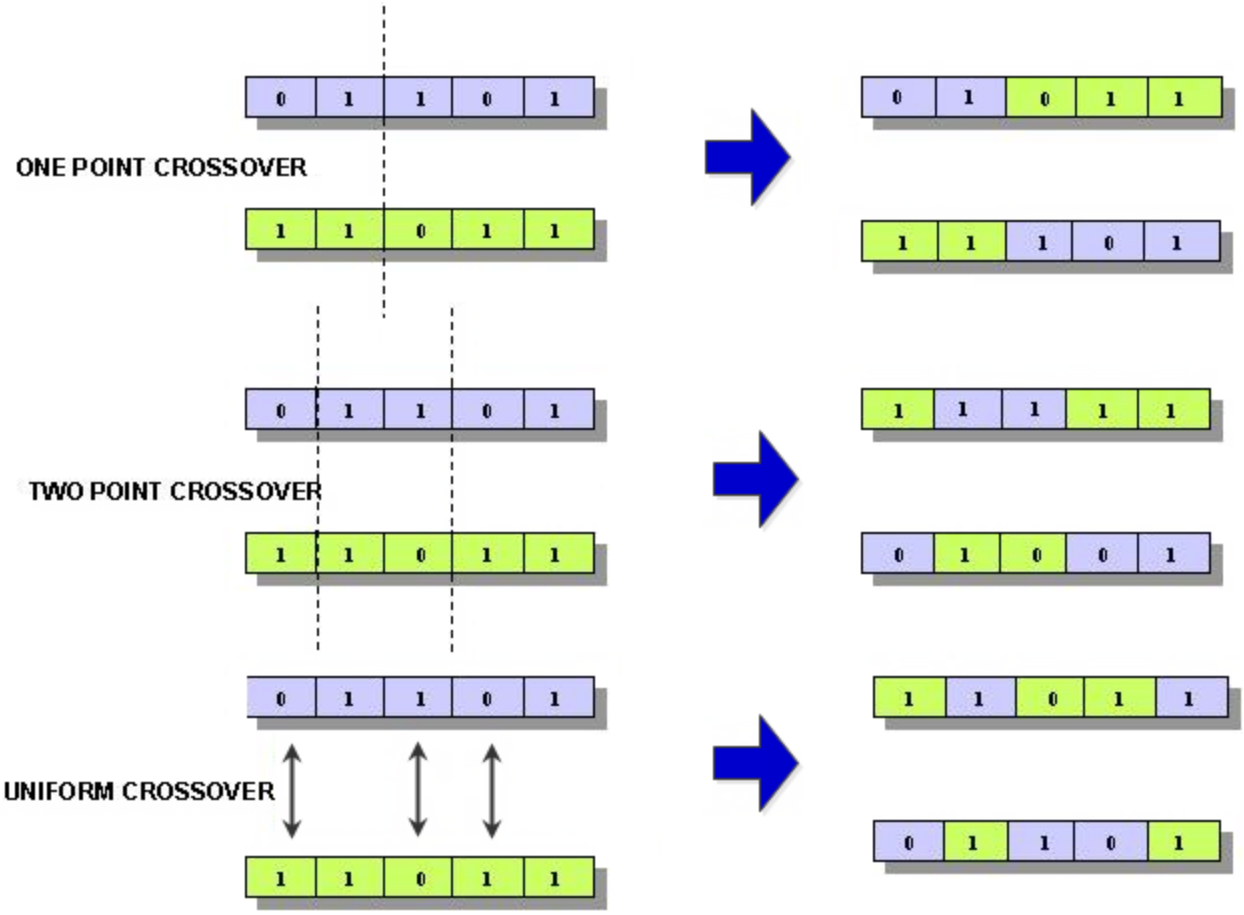
- Two-Point

  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.

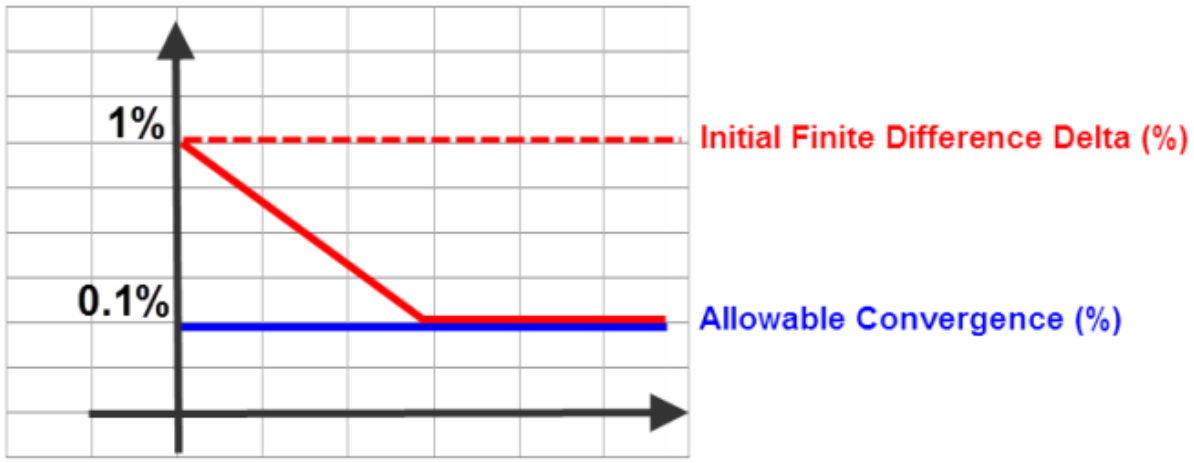
For NLPQL and MISQP, the default convergence rate, which is specified by the Allowable Convergence (%) property, is set to 0.1% for a Direct Optimization system. The maximum value for this property is 100%. This is computed based on the (normalized) Karush-Kuhn-Tucker (KKT) condition. This implies that 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.

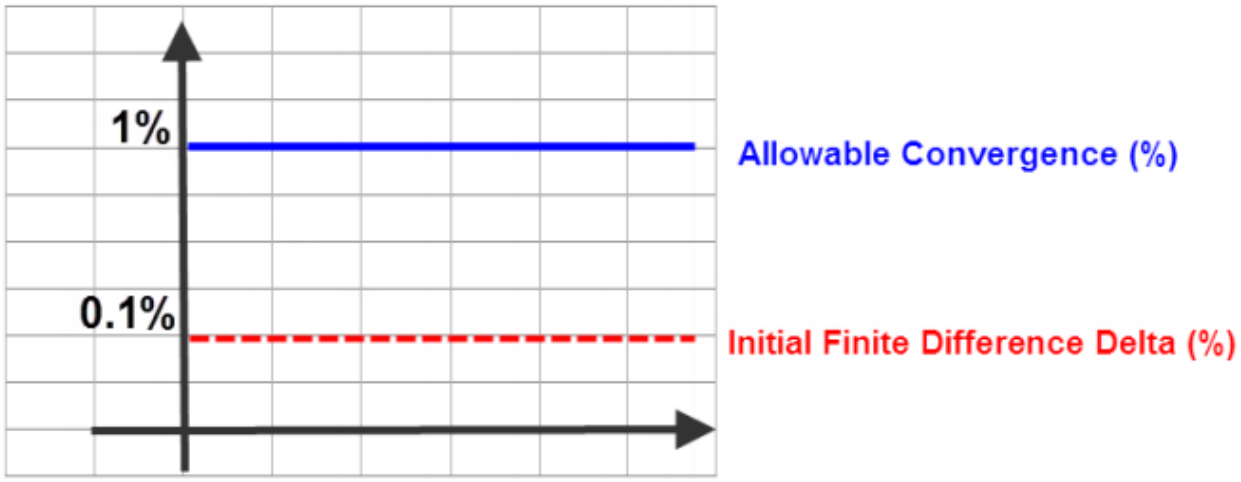
The advantage of this approach is that for large problems, it is possible to get a near-optimal feasible solution quickly without being trapped into a series of iterations involving small solution steps near the optima. To work most effectively with NLPQL and MISQP, keep the following guidelines in mind:

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

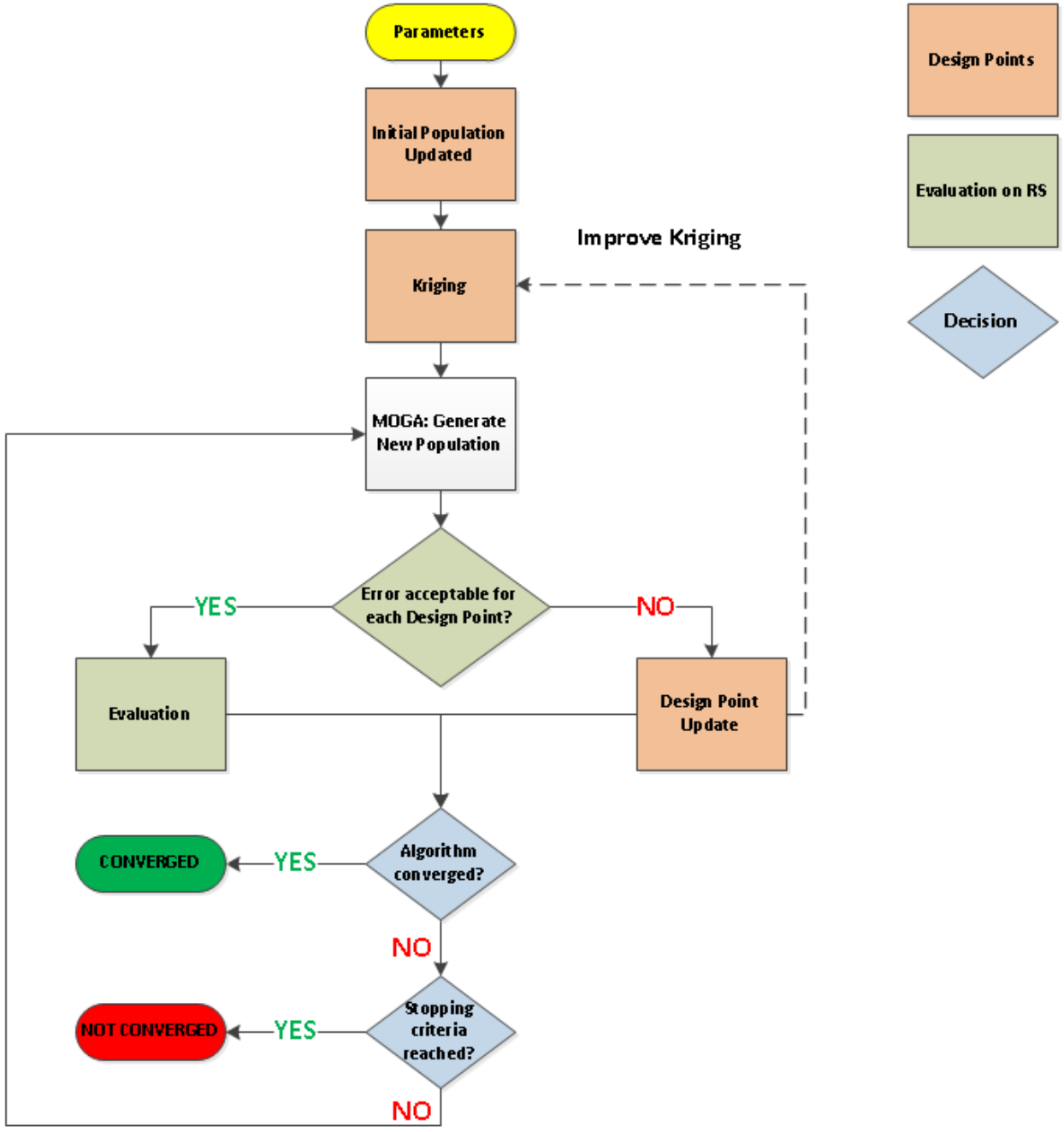
Adaptive Multiple-Objective (AMO) is a mathematical optimization that combines a Kriging response surface and MOGA. It allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. Except when necessary, the optimizer does not evaluate 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:

$$g_k(\{x\}) \leq 0, \forall k = 1,2,\ldots,K$$

$$h_l(\{x\}) = 0, \forall I = 1,2\ldots,L$$

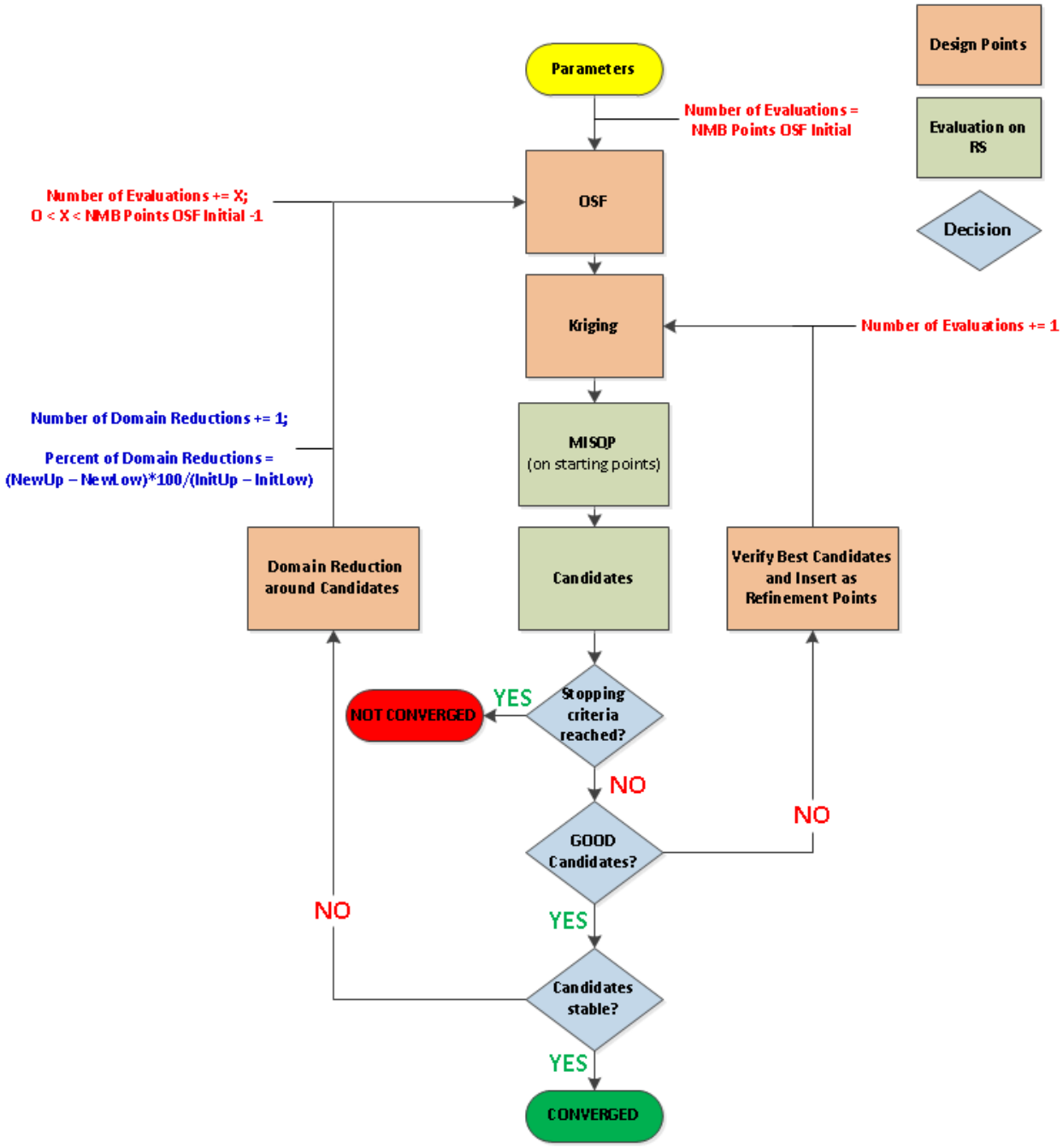
Where:

$$\{x_L\} \leq \{x\} \leq \{x_U\}$$

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.

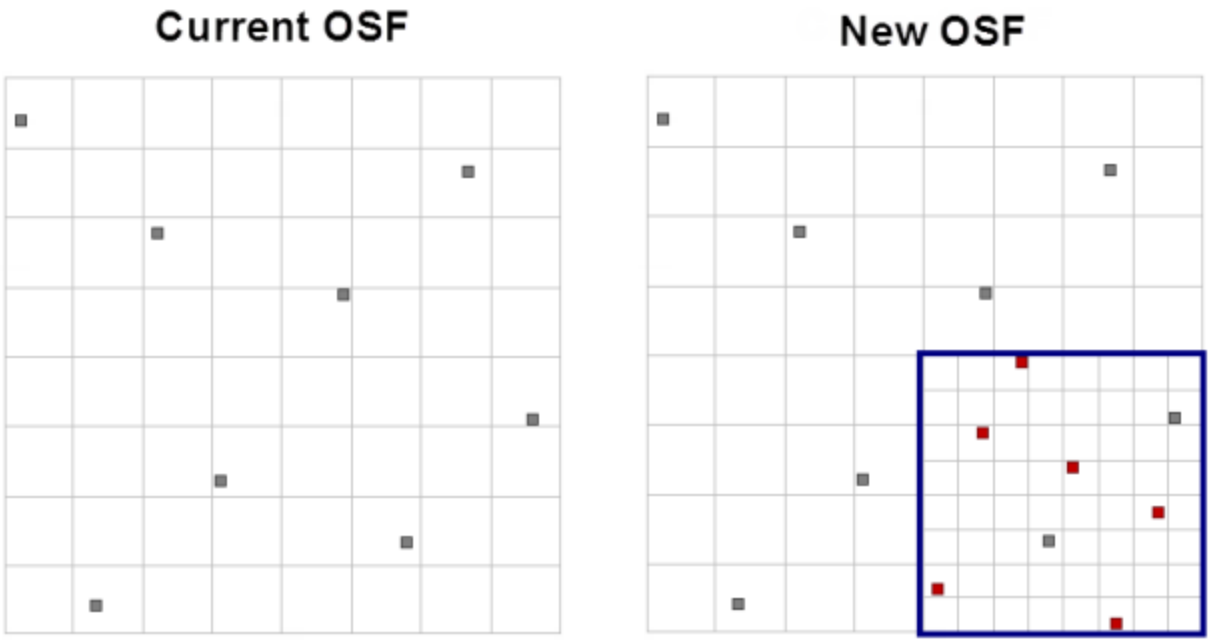
AMO Steps

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 (cos*t*(*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:

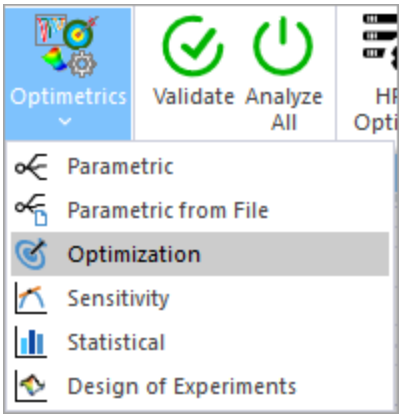
- Screening (Search based)
- Multi-Objective Genetic Algorithm
- Nonlinear Programming by Quadratic Lagrangian (Gradient)
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)
- Adaptive Multiple Objective (Gradient)
- Adaptive Single Objective
- MATLAB

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

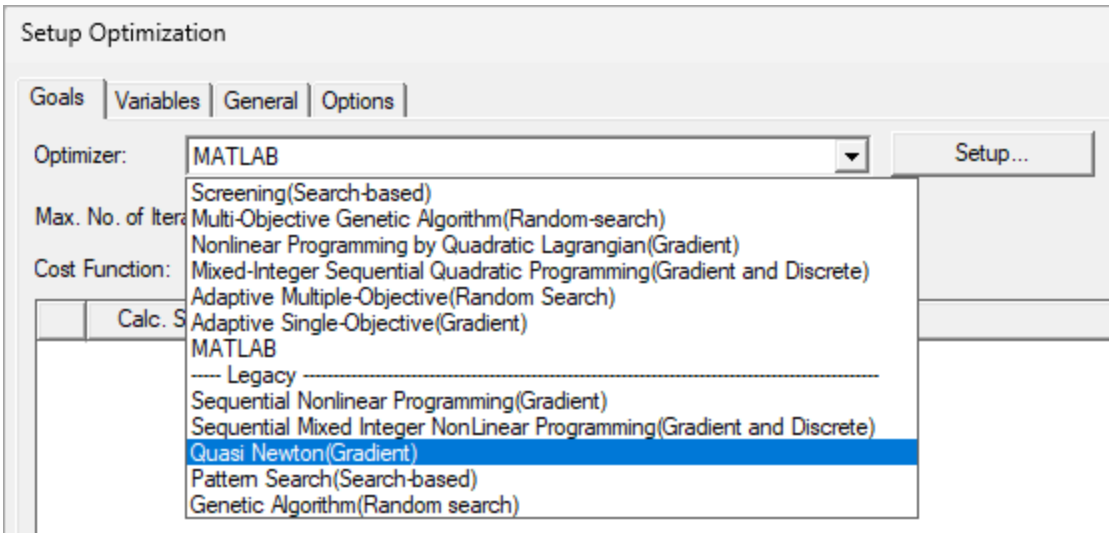
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **HFSS 3D Layout** > **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

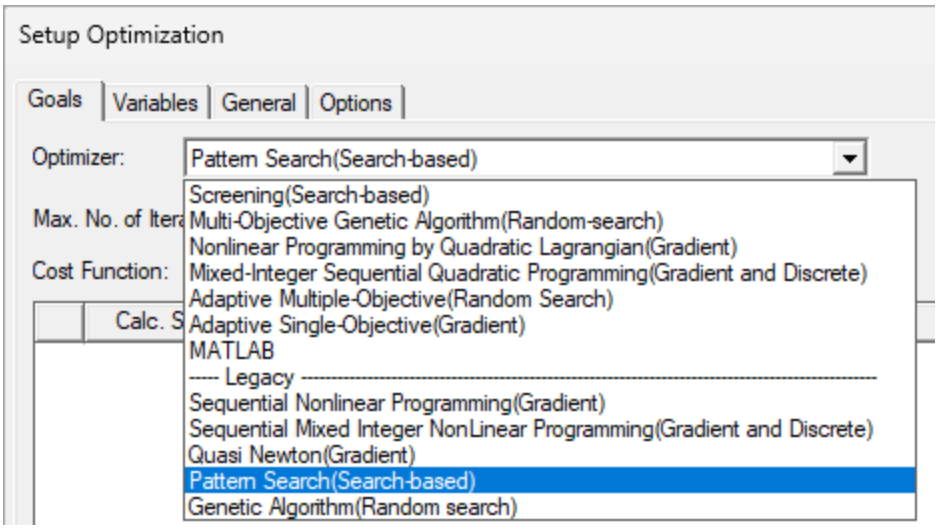
Following is the procedure for setting up an optimization analysis using the Pattern Search (Search-based) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and

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 **HFSS 3D Layout** > **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 **HFSS 3D Layout** > **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.

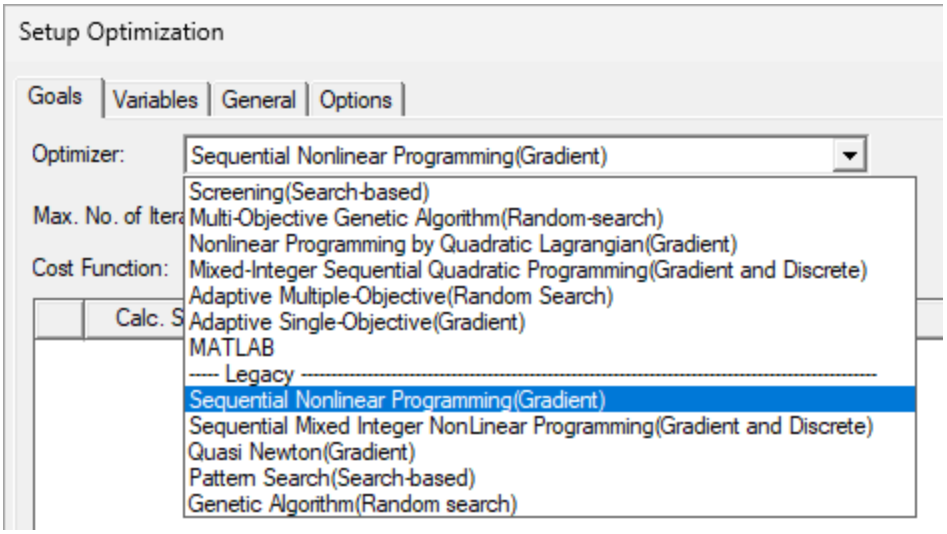
Setting Up the Sequential Nonlinear Programming (Gradient) Optimizer

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 **HFSS 3D Layout** > **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.

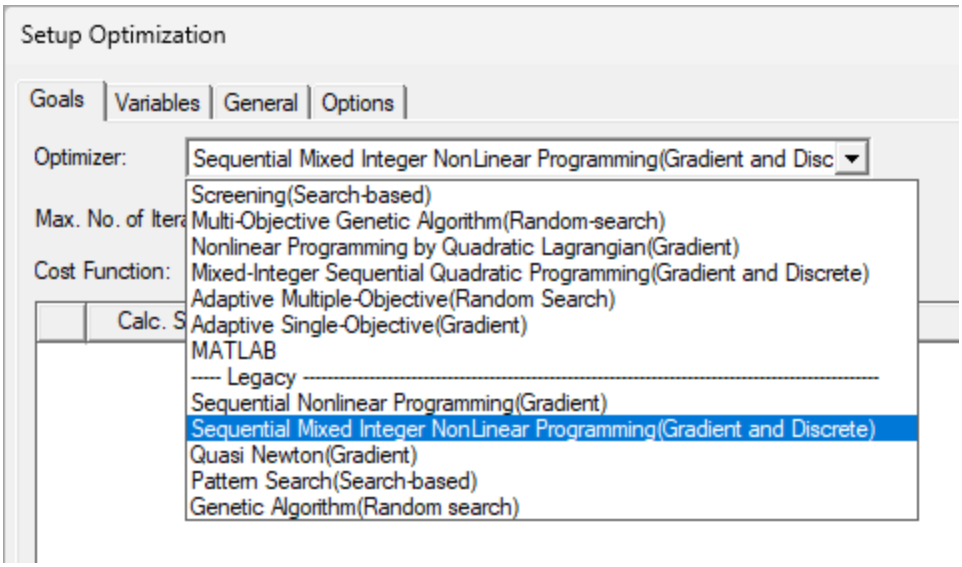
Setting Up the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer

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 **HFSS 3D Layout** > **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.

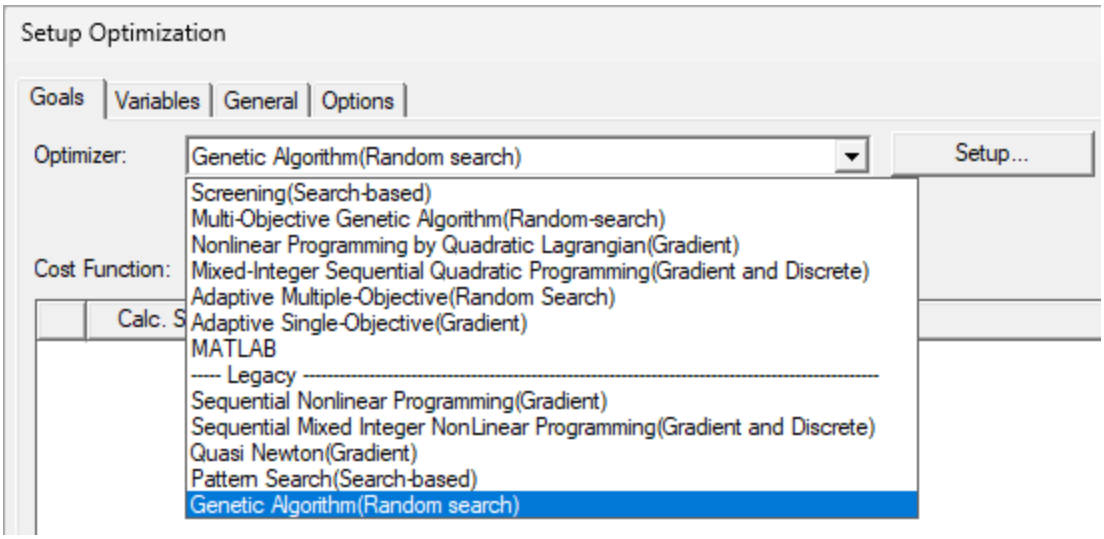
Setting Up the Genetic Algorithm (Random Search) Optimizer

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 **HFSS 3D Layout** > **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 **HFSS 3D Layout** > **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.

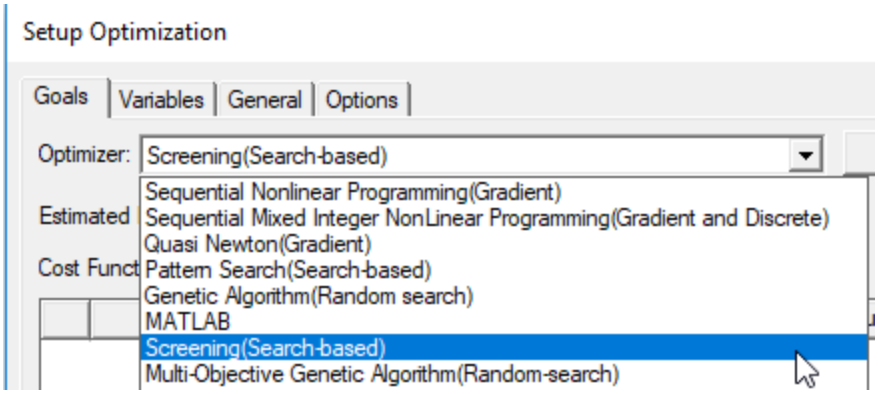
Screening (Search Based) Optimizer Setup

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.

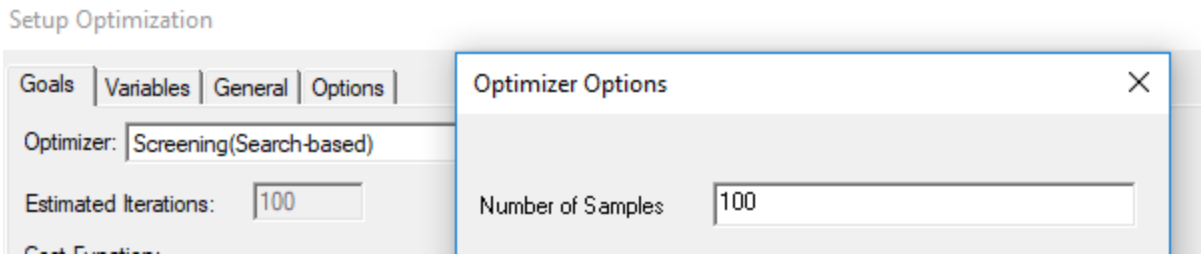
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **HFSS 3D Layout** > **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.



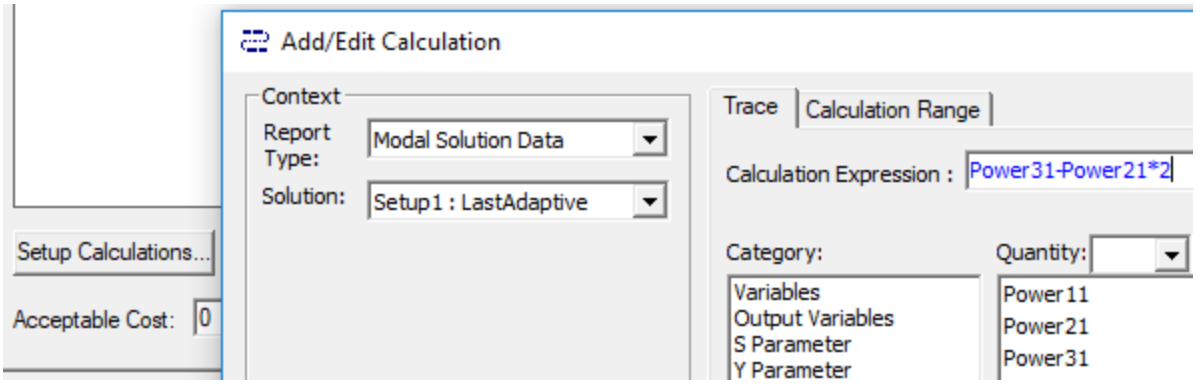
4. Optionally, press the **Setup** button to open the **Optimizer Options window** to change the default number of samples from 100.



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.

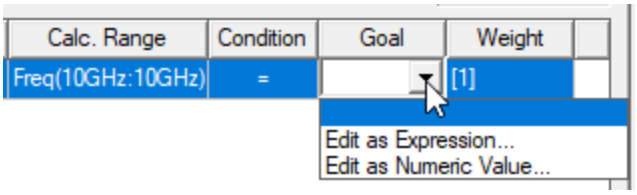
5. Add a cost function by selecting the **Setup Calculations** button to open the **Add/Edit Calculation** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

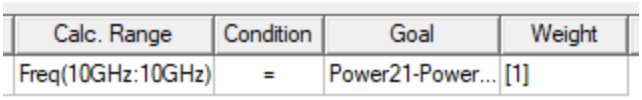


When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** window.

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

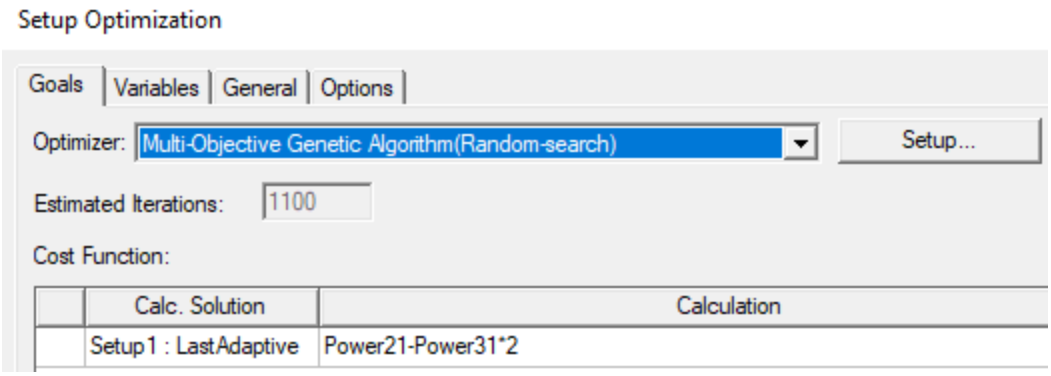
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

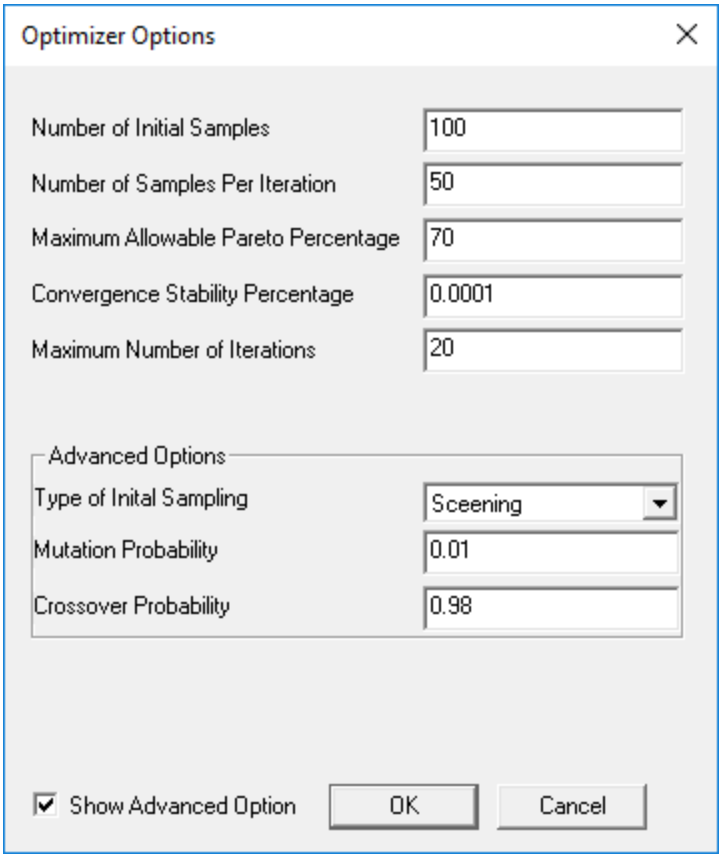
The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of 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 **HFSS 3D Layout** > **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

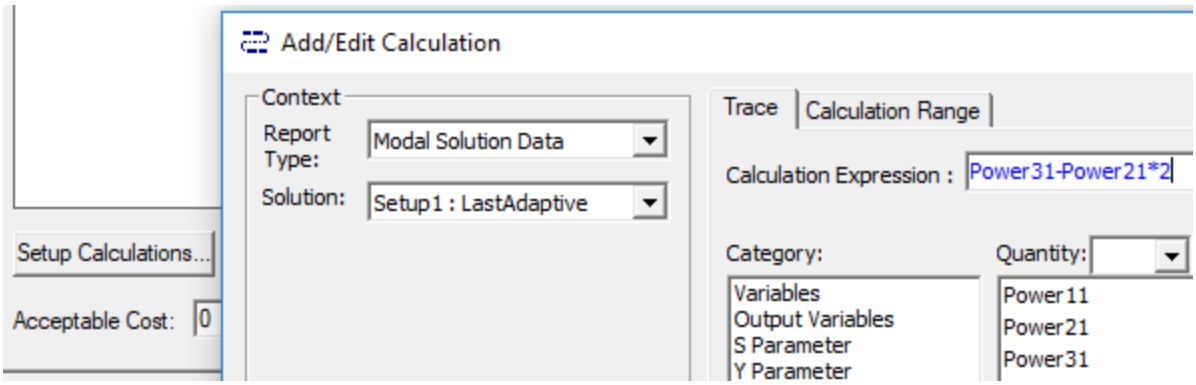
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **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.

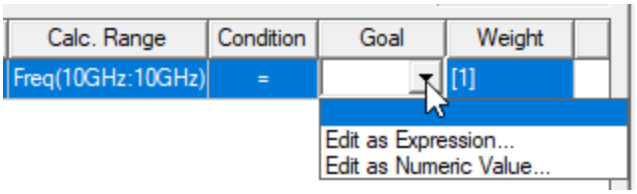
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the 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.

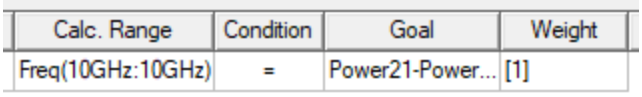
6. In the Optimization setup, in the drop-down menu for the Goal column, select either **Edit as Expression...** or **Edit as Numeric Value...**.



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

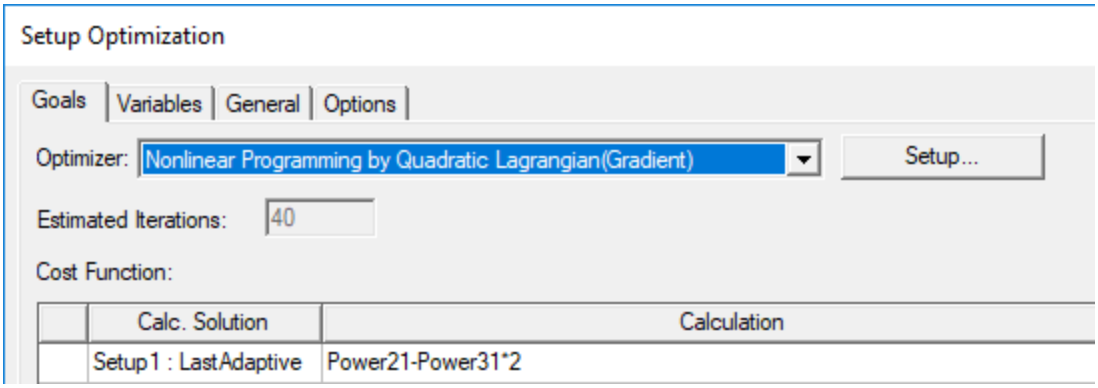
Following is the procedure for setting up an optimization analysis using the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

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).

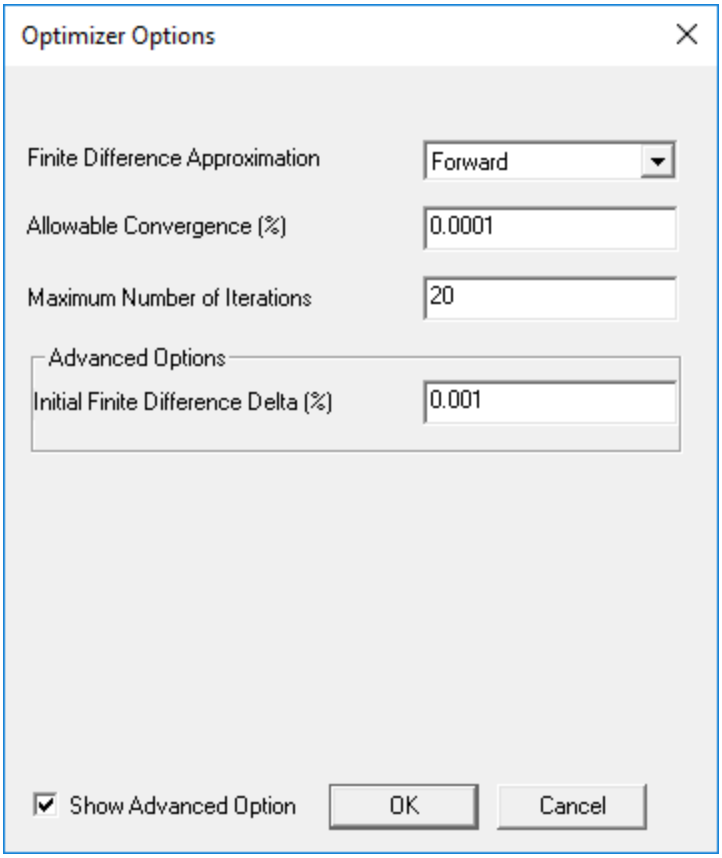
To generate samples and perform an NLPQL optimization:

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 **HFSS 3D Layout** > **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)

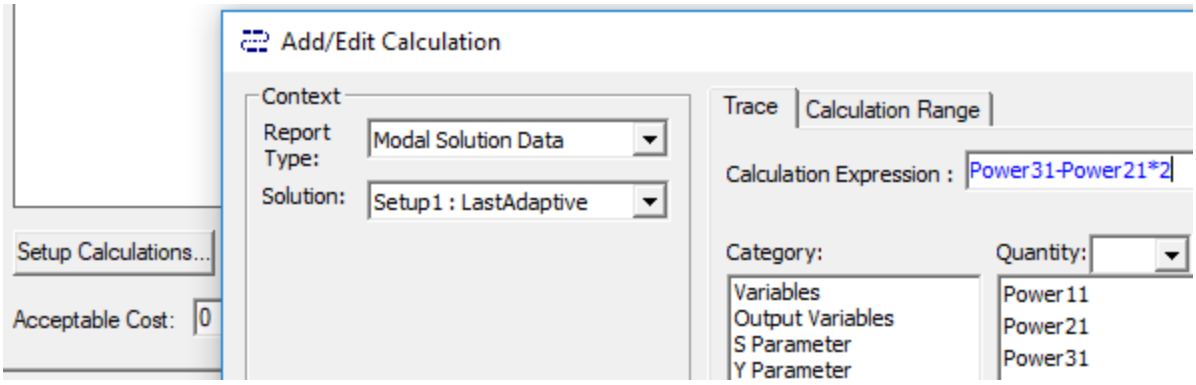
  For **Forward**: number of iterations * (number of inputs+1)

- **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.

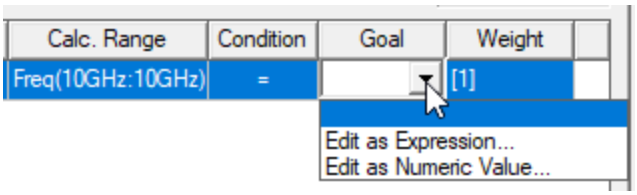
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



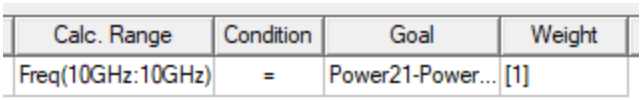
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

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.

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

$$g_j(x,y)=0,\ j=1,\ldots,m_e,$$
$$g_j(x,y)\geq 0,\ j=m_e+1,\ldots,m$$

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.

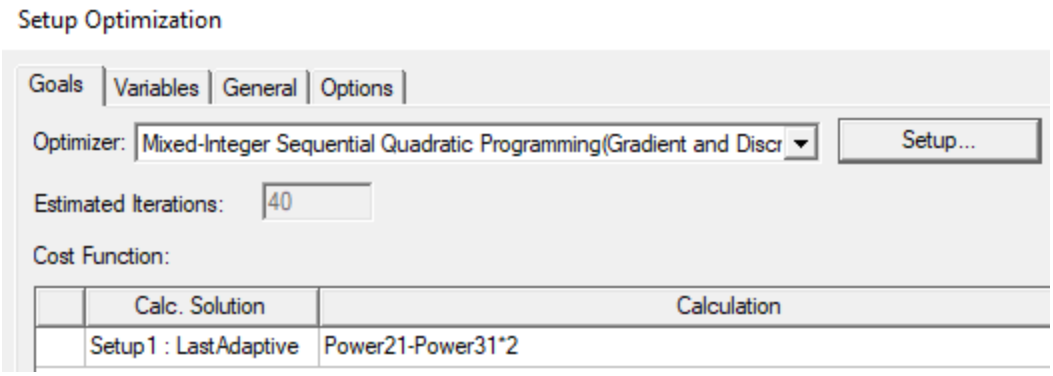
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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).

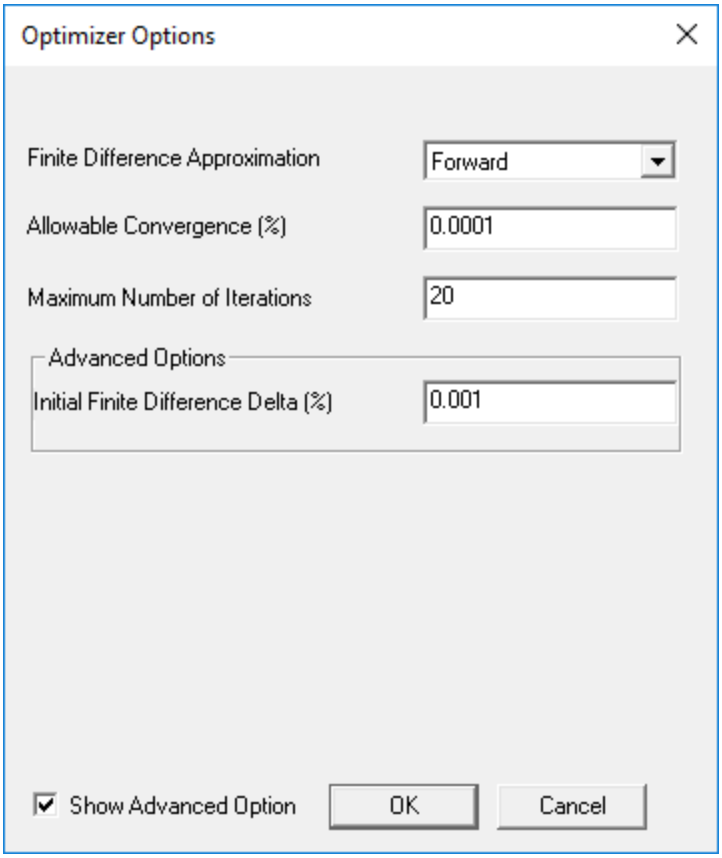
To generate samples and perform an NLPQL optimization:

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 **HFSS 3D Layout** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 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, MISQP approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For MISQP, the number of evaluations can be approximated according to the Finite Difference Approximation gradient calculation method, 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.

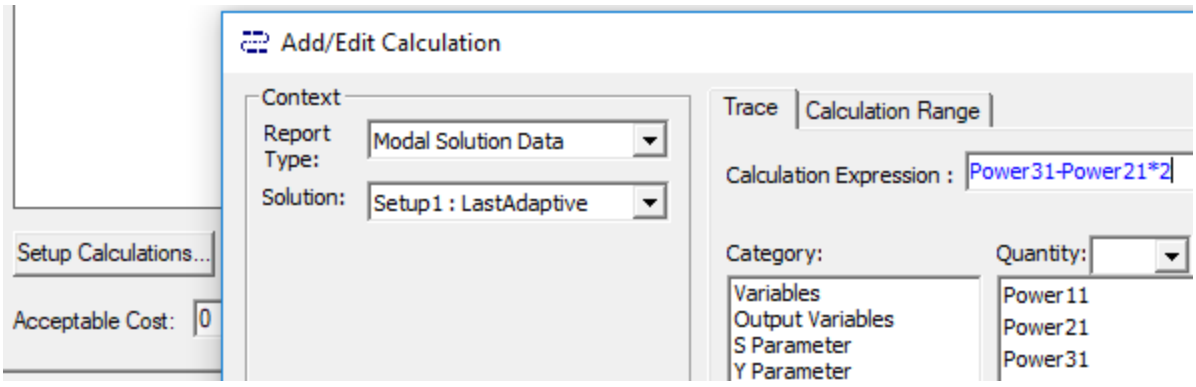
For **Central**: number of iterations * (2*number of inputs +1)

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.

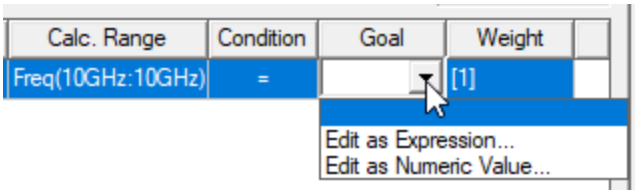
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

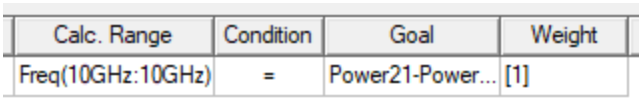
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the Optimization setup, in the drop-down menu, for the Goal column, select either Edit as Expression or Edit as Numeric Value...



   This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

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 and paste it, and then make changes to the copy, rather than redoing the whole process for minor changes.

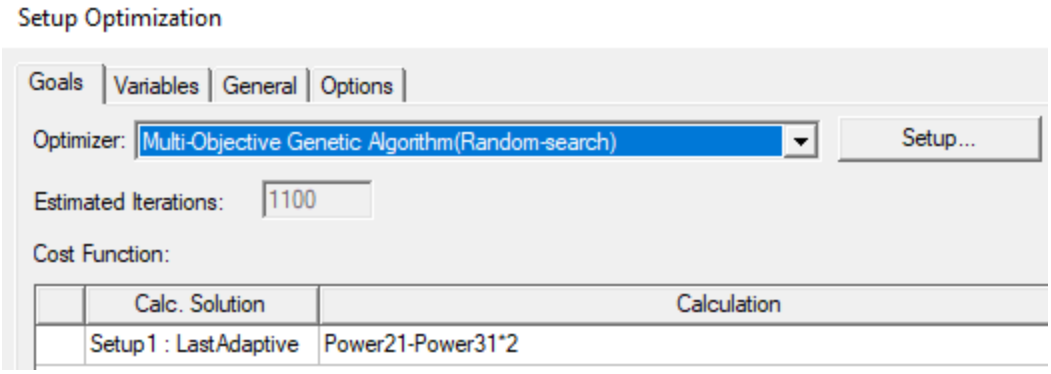
The Multi-Objective Genetic Algorithm (MOGA) is a hybrid variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports all types of input parameters. The Pareto ranking scheme is done by a fast, non-dominated sorting method that is an order 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

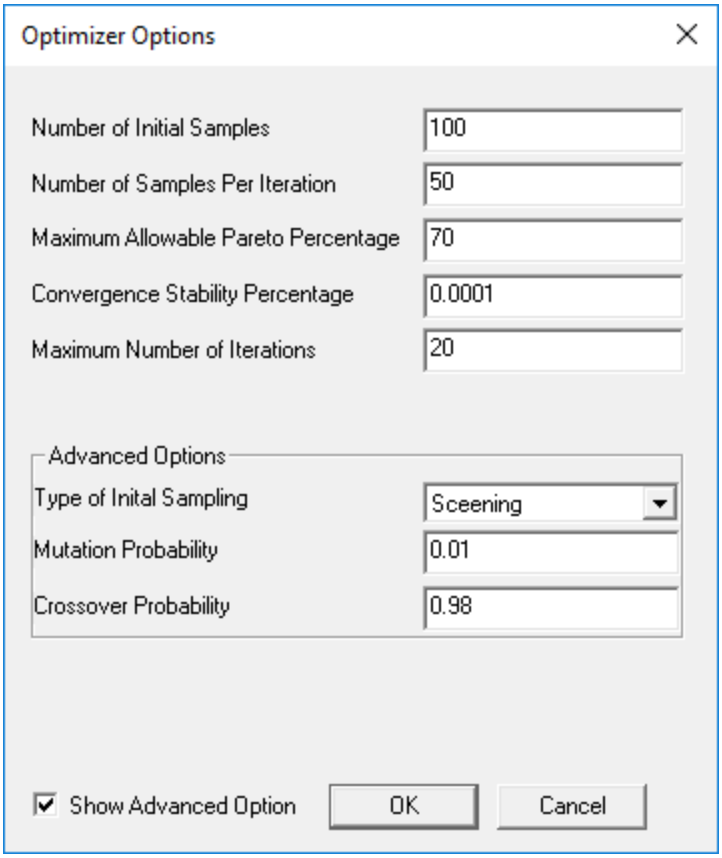
elitism of the process) to avoid premature convergence by altering the Maximum Allowable Pareto Percentage property.

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 **HFSS 3D Layout** > **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

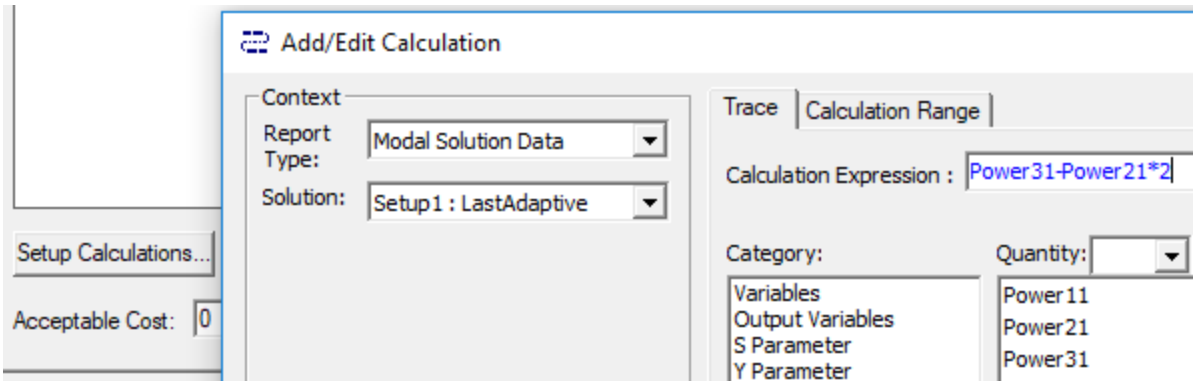
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **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.

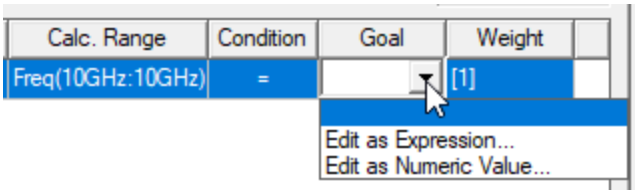
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the 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.

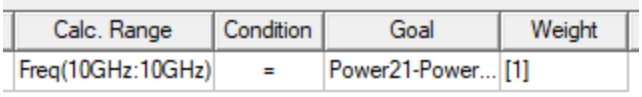
6. In the Optimization setup, in the drop-down menu for the Goal column, select either **Edit as Expression...** or **Edit as Numeric Value...**.



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.

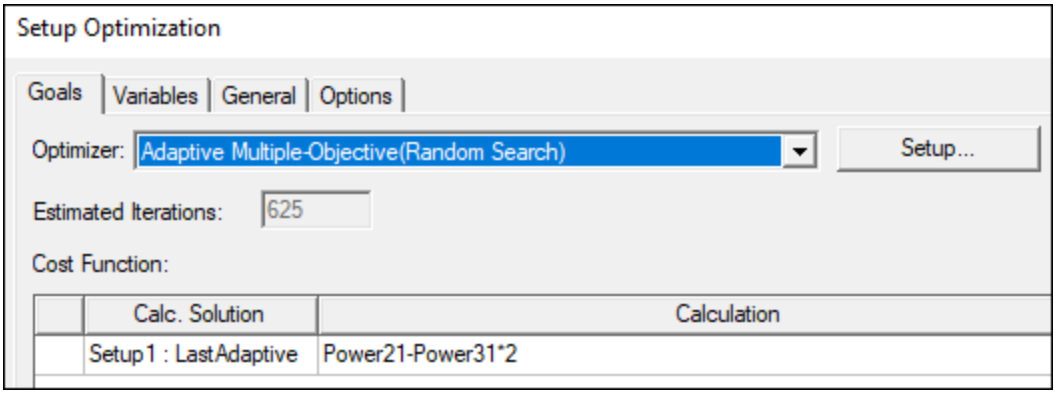
Adaptive Multiple-Objective (Random Search) Optimizer Setup

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.

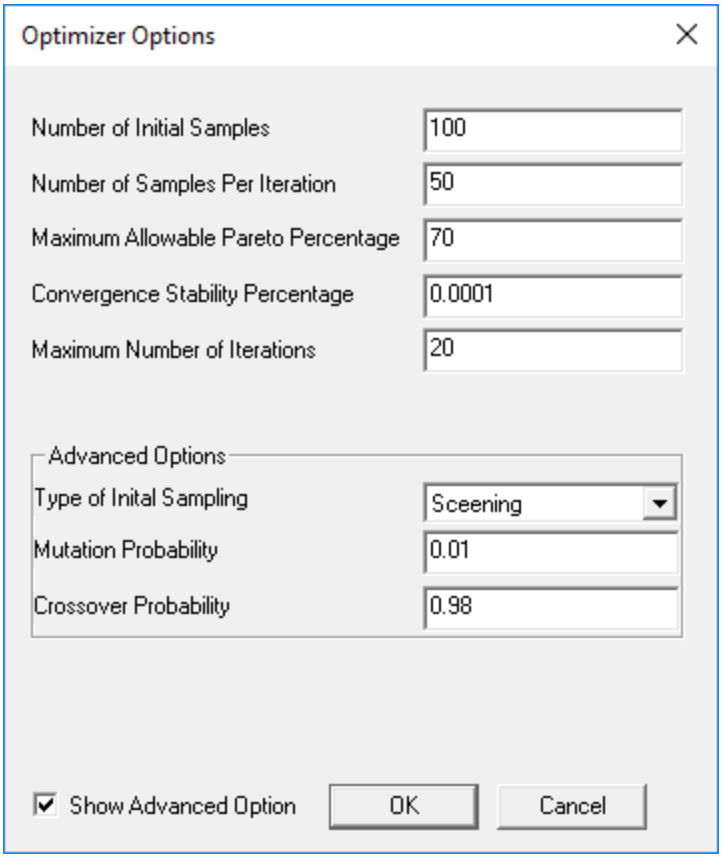
The Adaptive Multiple Objective (Kriging + MOGA) is an iterative algorithm that allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. It uses the same general approach as MOGA, but applies the Kriging error predictor to reduce 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 **HFSS 3D Layout** > **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.

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 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.

  You can enter a minimum of 2 and a maximum of 10000.

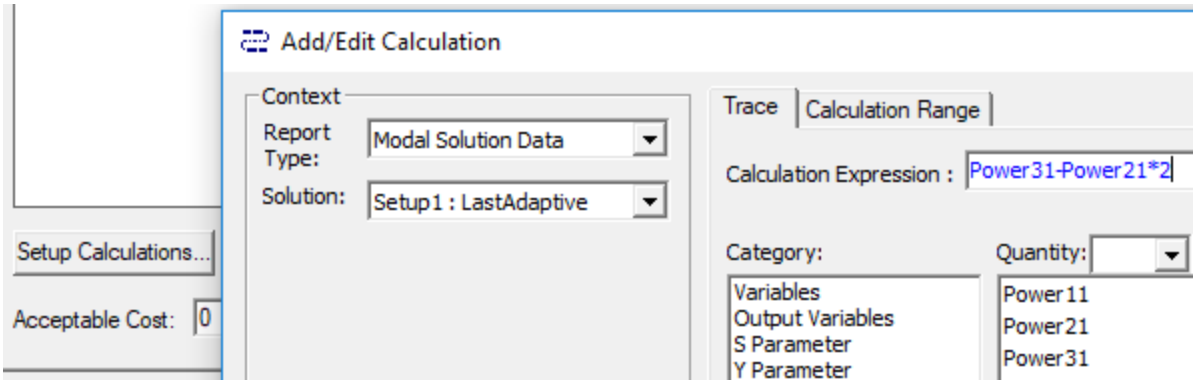
- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples 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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.
- **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.

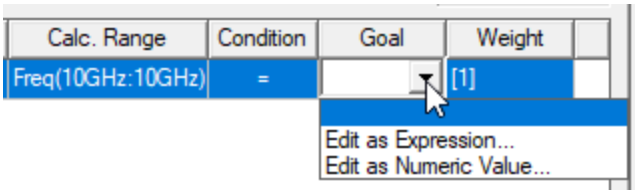
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



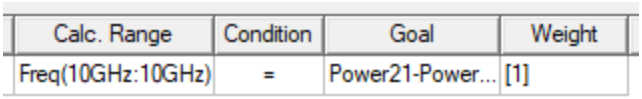
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

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.

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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

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.

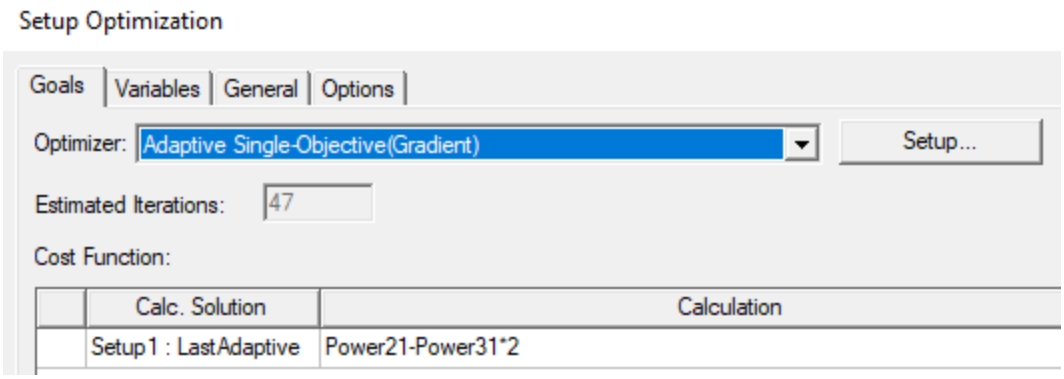
This gradient-based method employs automatic intelligent refinement to provide the global optima. It requires a minimum number of design points to build the Kriging response surface, but, in general, this method reduces the number of design points necessary for the optimization. Failed design points 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.

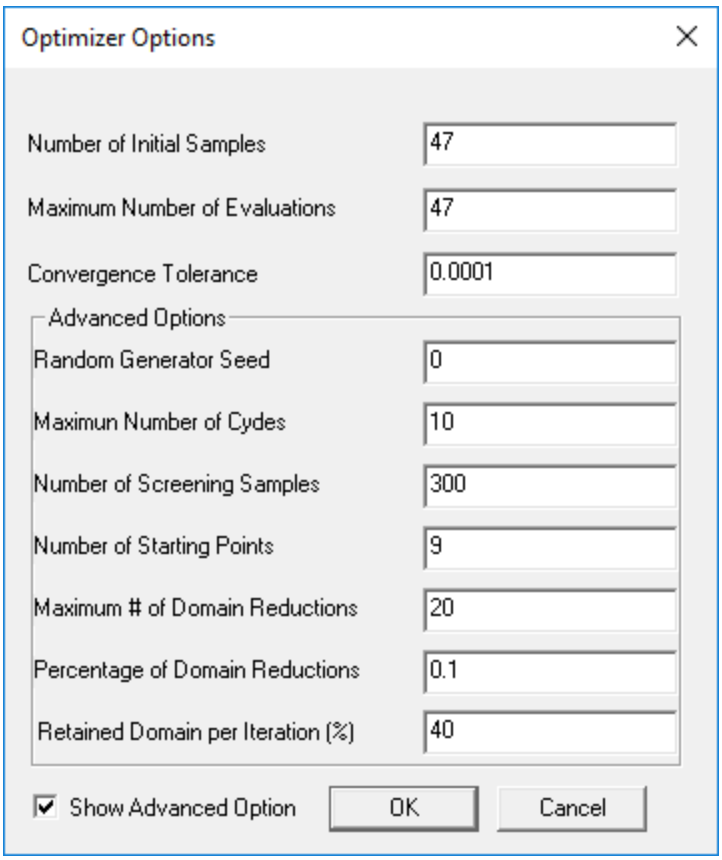
1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **HFSS 3D Layout** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under the **Goals** tab, choose the optimizer by selecting **Adaptive Single Objective (Gradient)** from the **Optimizer** drop-down menu.



4. Optionally, press the **Setup** button to open the *Optimizer Options* window and check **Advanced Options**.



- **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default is (NbInp+1)*(NbInp+2)/2.

Because of the Adaptive Single-Objective workflow (in which a new OSF sample set is generated after each domain reduction), increasing the number of OSF samples does not necessarily improve the quality of the results and significantly increases the number of evaluations

- **Maximum Number of Evaluations**: Stop criterion. Maximum number of evaluations (design points) that the algorithm is to calculate. If convergence occurs before this number is reached, evaluations stop. This value also provides an idea of the maximum possible time it takes to run the optimization. The default is 20*(NbInp +1).
- **Convergence Tolerance**: Stop criterion. Minimum allowable gap between the values of two successive candidates. If the difference between two successive candidates is smaller than the value for Convergence Tolerance multiplied by the maximum variation of the parameter, the algorithm is stopped. A smaller value 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.

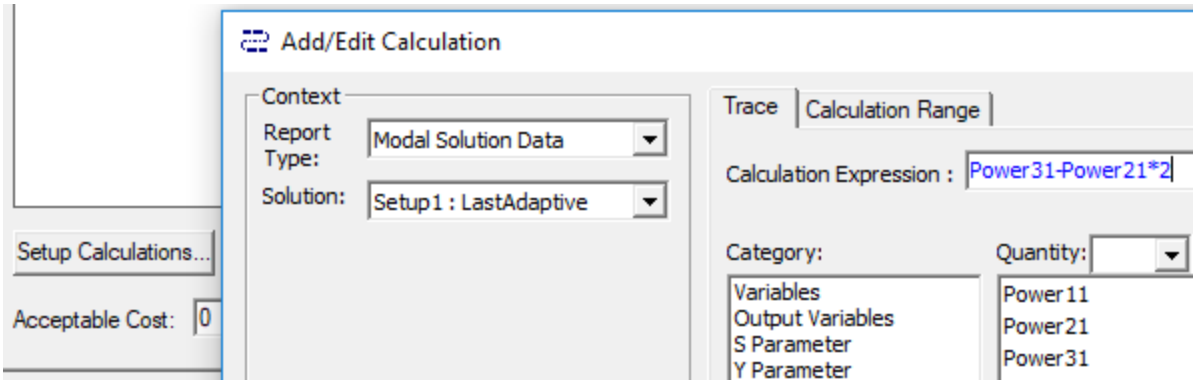
- **Number of Starting Points**: Determines the number of local optima to explore. The larger the number of starting points, the more local optima explored. In the case

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of a linear surface, for example, it is not necessary to use many points. This value must be less than the value for **Number of Screening Samples** because these samples are selected in this sample. The default is the value for **Number of Initial Samples**.

- **Maximum Number of Domain Reductions**: Stop criterion. Maximum number of domain reductions for input variation. (No information is known about the size of the reduction beforehand.) The default is 20.
- **Percentage of Domain Reductions**: Stop criterion. Minimum size of the current domain according to the initial domain. For example, with one input ranging between 0 and 100, the domain size is equal to 100. The percentage of domain reduction is 1%, so the current working domain size cannot be less than 1 (such as an input ranging between 5 and 6). The default is 0.1.
- **Retained Domain per Iteration (%)**: Advanced option that allows you to specify the minimum percentage of the domain you want to keep after a domain reduction. The percentage value must be between 10 and 90. A larger value indicates less domain reduction, which implies better exploration but a slower solution. A smaller value indicates a faster and more accurate solution, with the risk of it being a local one. The default percentage value is 40.

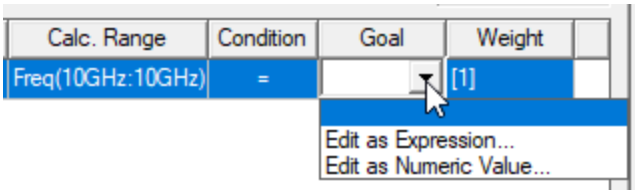
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



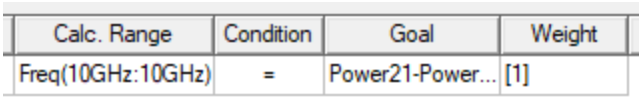
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog box.

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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

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.

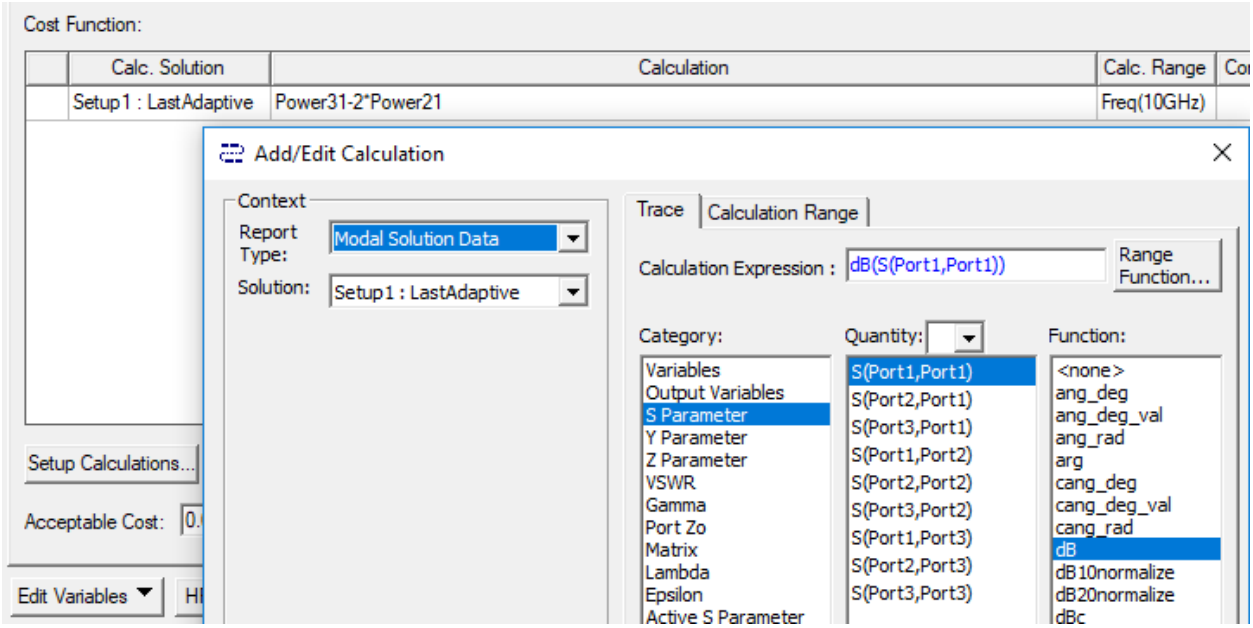
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

$$c_{perturbed} = 10000 \cdot (|S_{11}|_{min} + E_{S11})^2$$

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

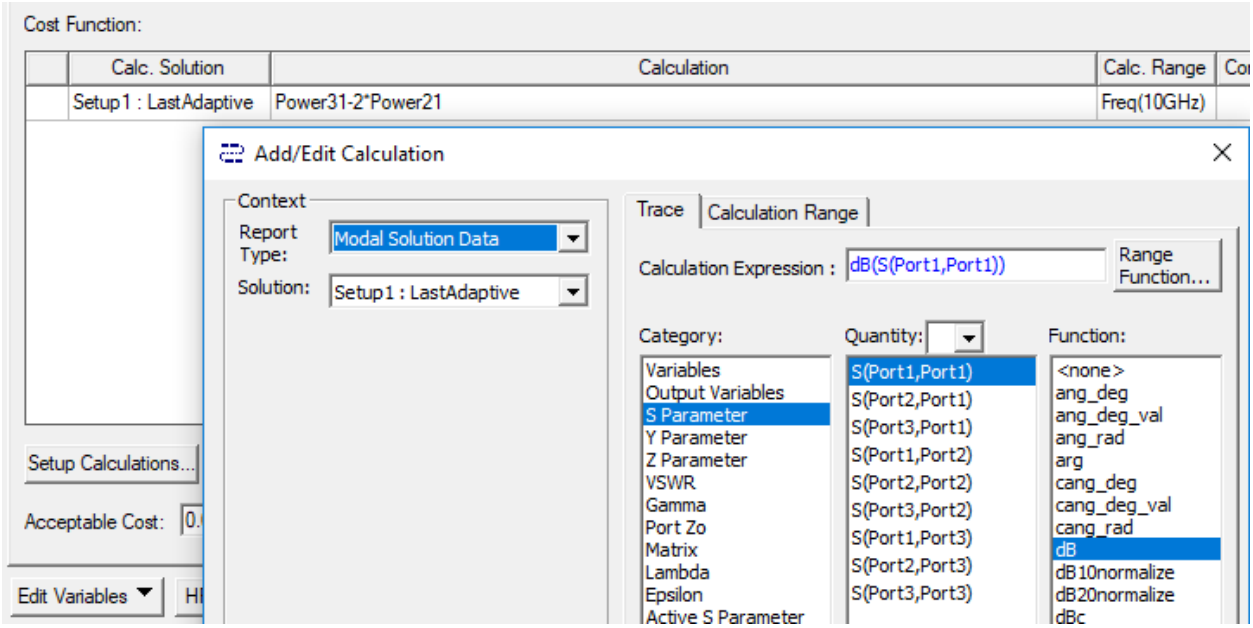
A cost function can include one or more goals for an optimization analysis. Optimetrics manipulates the model's design variable values to fulfill the cost function. The optimization will stop when the solution quantity meets the acceptable cost criterion.

Following is the general procedure for adding a cost function with a single goal:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

   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.

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 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.

     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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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 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.

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.

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 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

- 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	**e_i evaluation where s_i is the simulated response**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Condition)	and g_i is the desired limit.
<=	$e_i = \begin{cases} 0 \\ s_i - g_i \end{cases}$ $s_i \leq g_i$ $s_i > g_i$
=	$e_i = \lvert s_i - g_i \rvert$
>=	$e_i = \begin{cases} 0 \\ g_i - s_i \end{cases}$ $s_i \geq g_i$ $s_i < g_i$

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.

> **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.

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 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

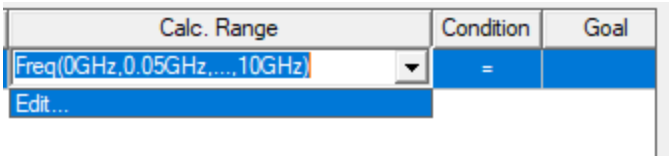
For every optimization setup, Optimetrics automatically sets the minimum and maximum values it will consider for a variable being optimized. Optimetrics sets a variable's minimum value equal to approximately 50% of its starting value. (The starting value is the variable's current value set for the nominal 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.

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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

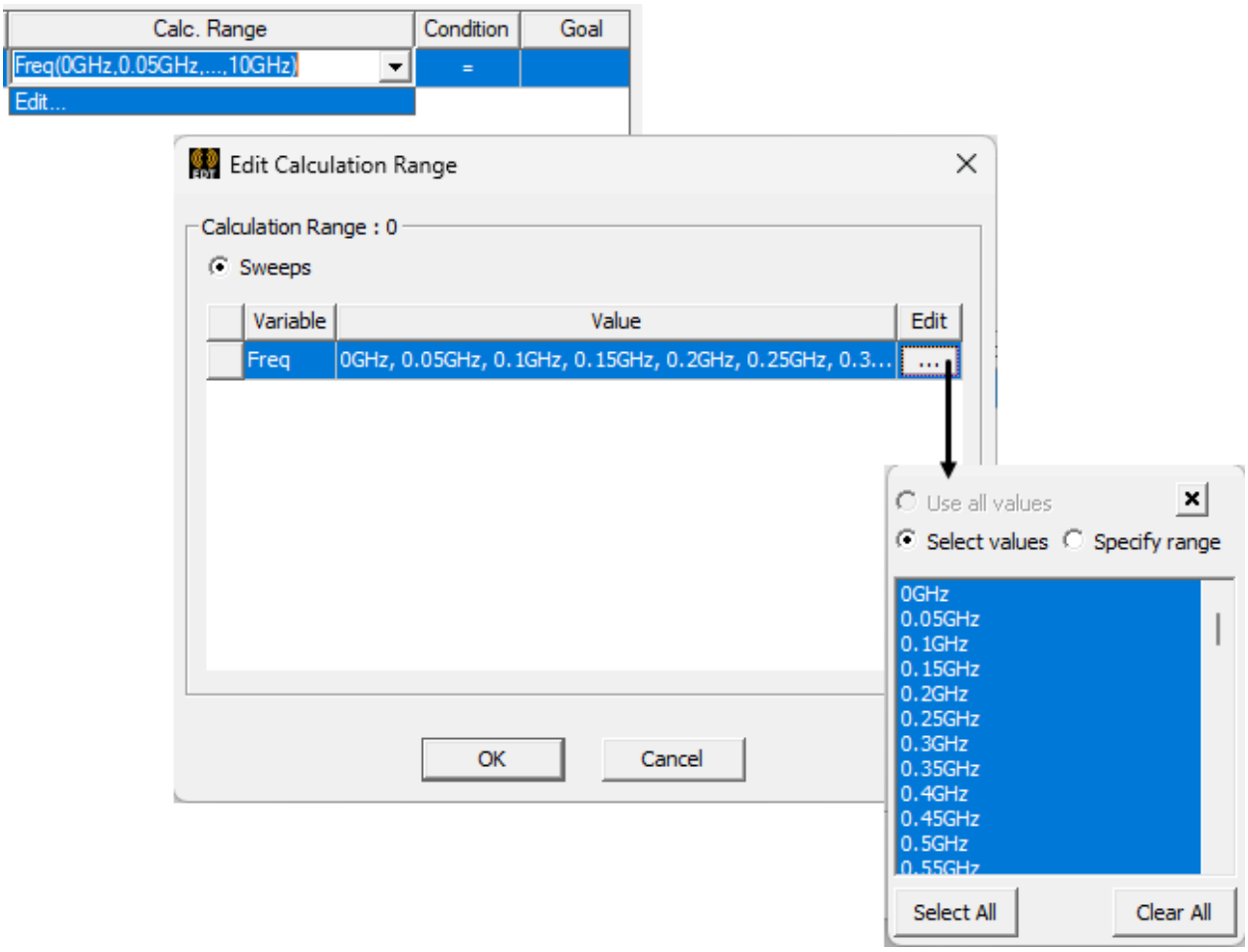
- You solve 0 to 10 GHz with a step value of 0.05.
- You can specify multiple sweep values by separating those with comma ","

  For example, F(1GHz), cap(1pf, 1.2pf)

  For example, Distance(All), Freq([1ghz,2ghz]), Phase(0 deg)

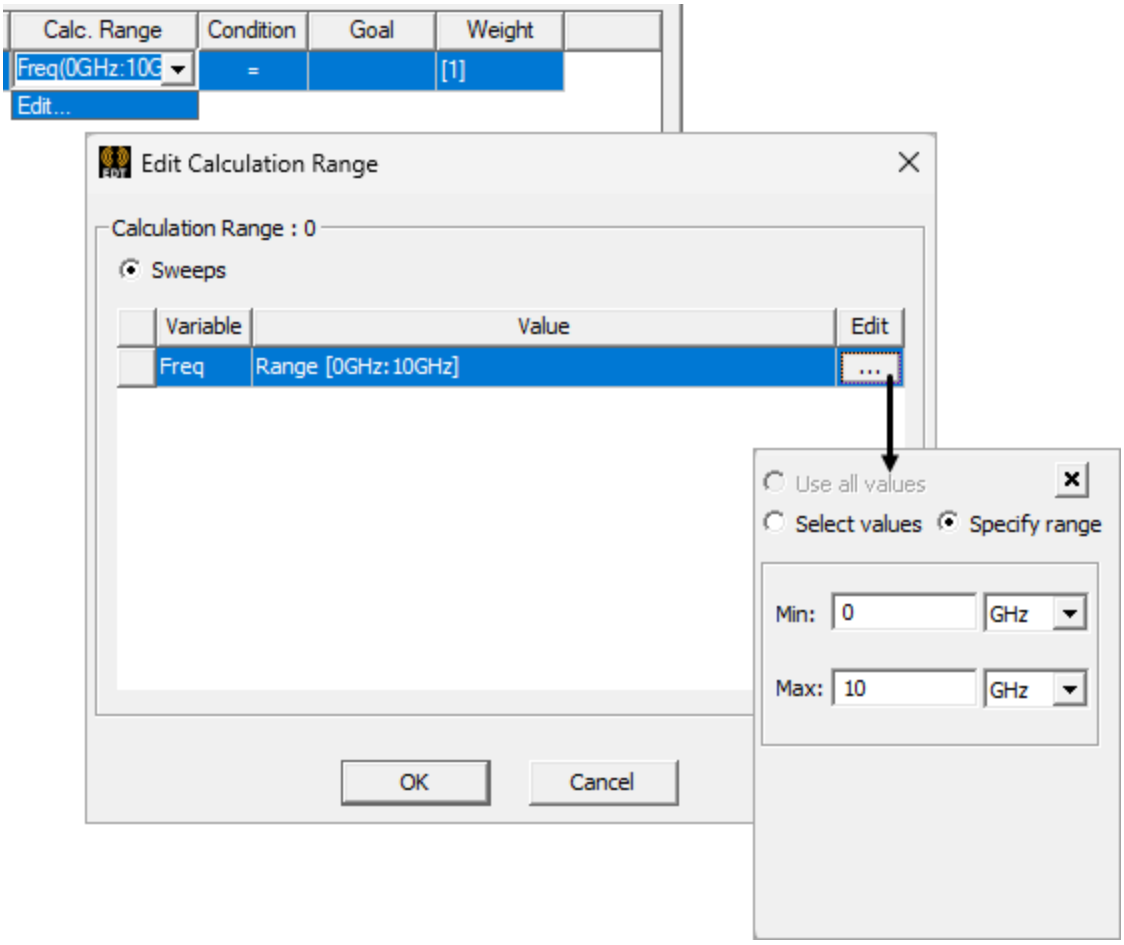
If you click **Edit..** on the menu, you see the **Edit Calculation Range** dialog box. Click on the ellipsis [...] button to select radio buttons for Use all values, Select Values or Specify range.

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 **HFSS 3D Layout**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

   The *Properties* dialog box appears.
3. Select **Optimization**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.
5. Click **OK**.

   When Optimetrics solves an optimization setup, it does not consider variable values that lie outside of this range.

Step Size

To make the search for the minimum cost value reasonable, the search algorithm is limited in two ways. First, you do not want the optimizer to continue the search if the step size becomes irrelevant or small. This limitation impacts the accuracy of the final optimum. Second, in some cases you do not want the optimizer to take large steps either. In case the cost function is suspected to possess large variations in a relatively small vicinity of the design space, large steps may result in too many trial steps, which do not improve the cost value. In these cases, it is safer to proceed with limited size steps and have more frequent improvements.

For these two limitations, the optimizer uses two independent distance measures. Both are based on user-defined quantities: the minimum and maximum step limits for individual optimization variables. Since the particular step is in a general direction, these measures are combined together in order to derive the limitation for that particular direction.

The step vector between the i^{th} and $(i+1)^{th}$ iterate is as follows:

$$s_i = x_{i+1} - x_i$$

The natural distance measure is

$$\|s_i\| = \sqrt{s_i^T s_i}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

which is the Euclidean norm.

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.

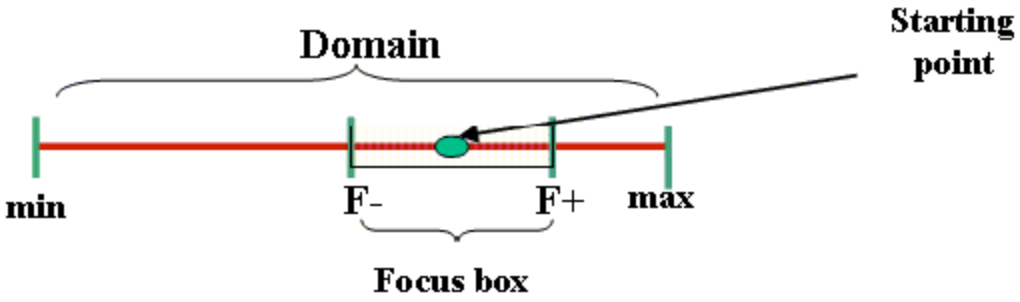
1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min Step** and **Max Step** columns, with default values for each variable to be optimized.
3. In the **Min Step** text box, type the minimum step size value. Optionally, modify the unit system in the **Units** 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.

$$Cost = \sum_{1}^{N} w_i \varepsilon_i^2$$

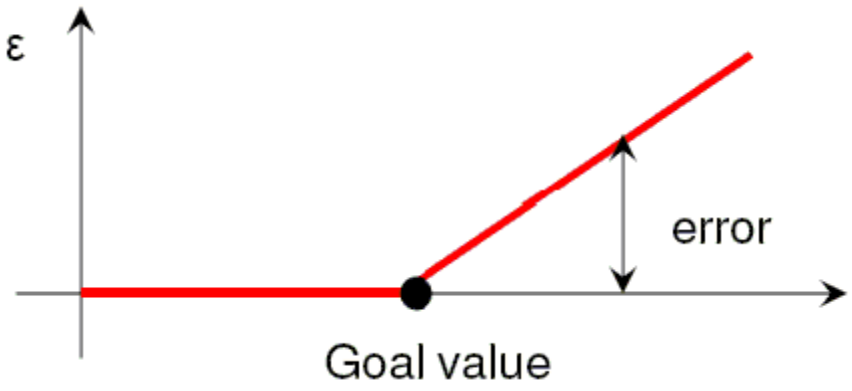
For the **Maximum** norm, the cost function uses the maximum among all the weighted goal errors, which means that cost is always less than zero:

$$Cost = \max_{1}^{N} W_i \cdot \varepsilon_i$$

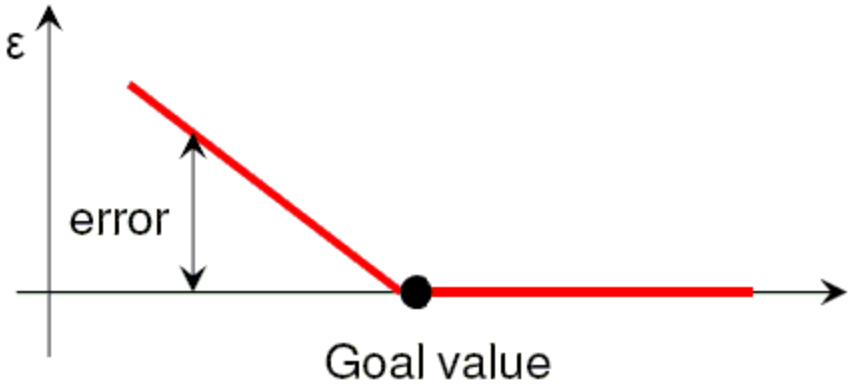
For all the above situations, N is the number of individual goals $w_i \varepsilon_i$ are individual weighting factors and residual error respectively. A minimization of the cost function is performed during optimization since it makes sense to minimize the error in the sense of the chosen norm type.

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:

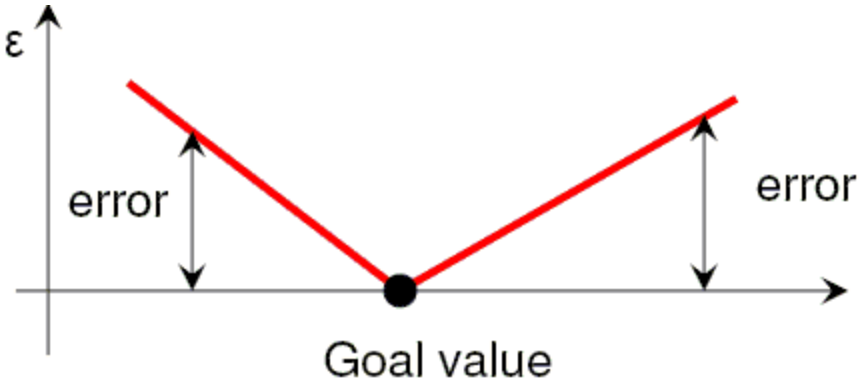
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:



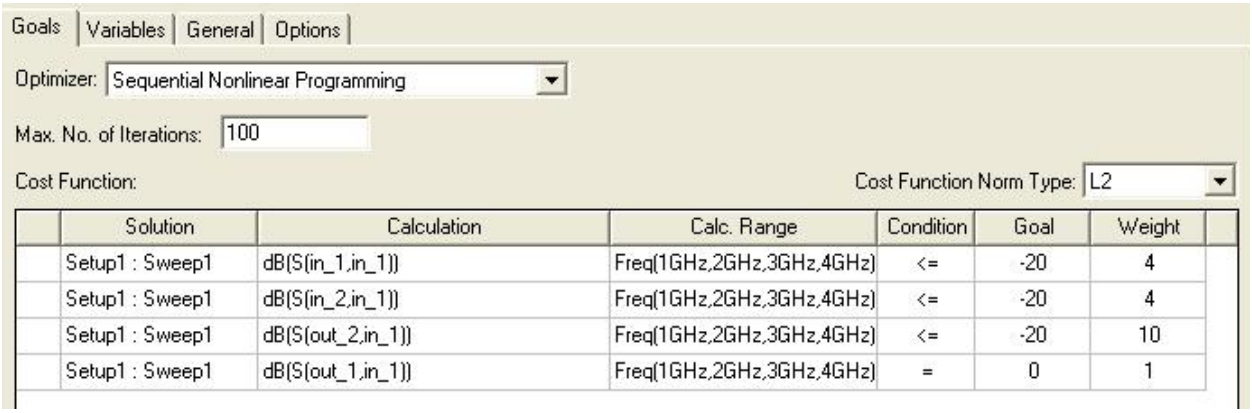
The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios. Note that when using "minimize" or "maximize" settings for the condition there should be a single goal setting which in this case coincides with the cost function.

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

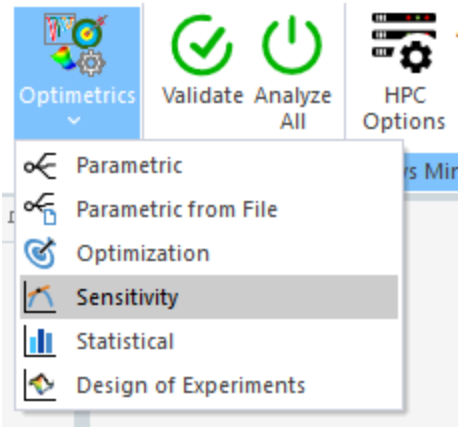
During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based 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 **HFSS 3D Layout** > **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 **HFSS 3D Layout**> **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:**

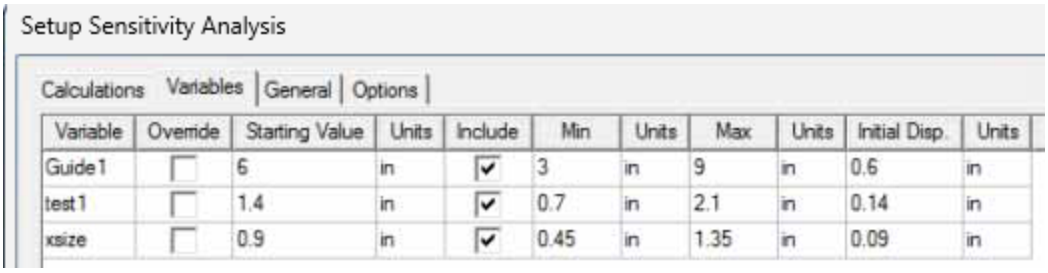
   This is one of the most popular methods to estimate worst-case performance. To start, a sensitivity analysis is performed. The results (sensitivities/first derivative) allow us to pick an extreme value (upper or lower bound) for each variable. The corresponding simulation result is used to predict upper 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**.

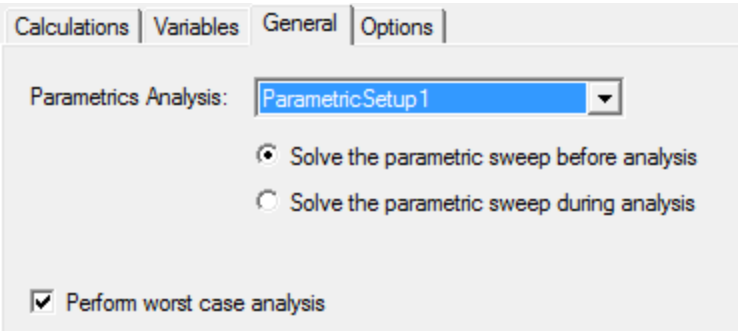
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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

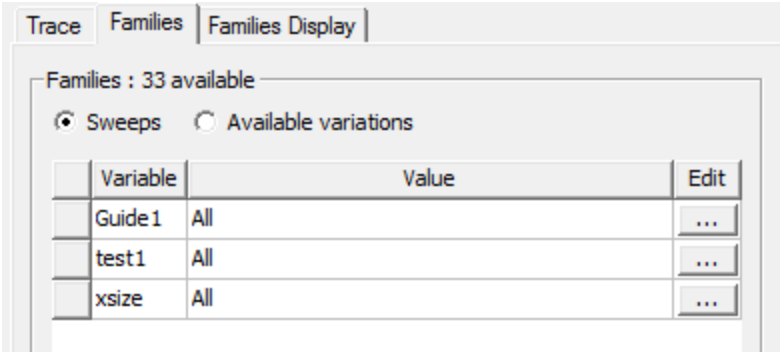
- After setup, analyze your Sensitivity Setup under Optimetrics.
- To view worst-case result (upper bound only):

  Create a Data Table report.

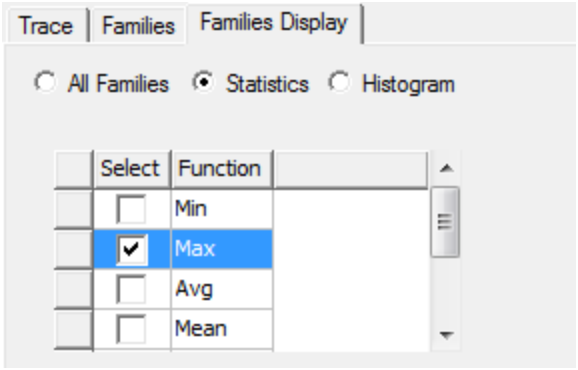
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

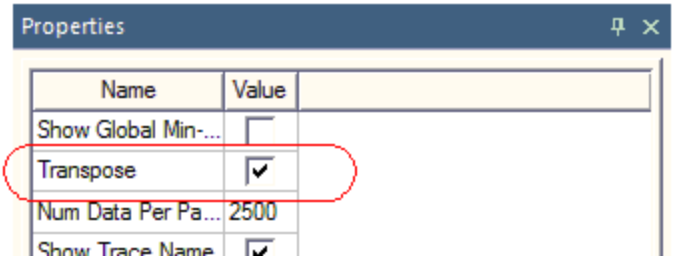
	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Max
1	3.000000	-0.000000
2	4.000000	-0.000000
3	5.000000	-0.105323
4	6.000000	-0.248322
5	7.000000	-0.267220
6	8.000000	-0.202039
7	9.000000	-0.298840

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:

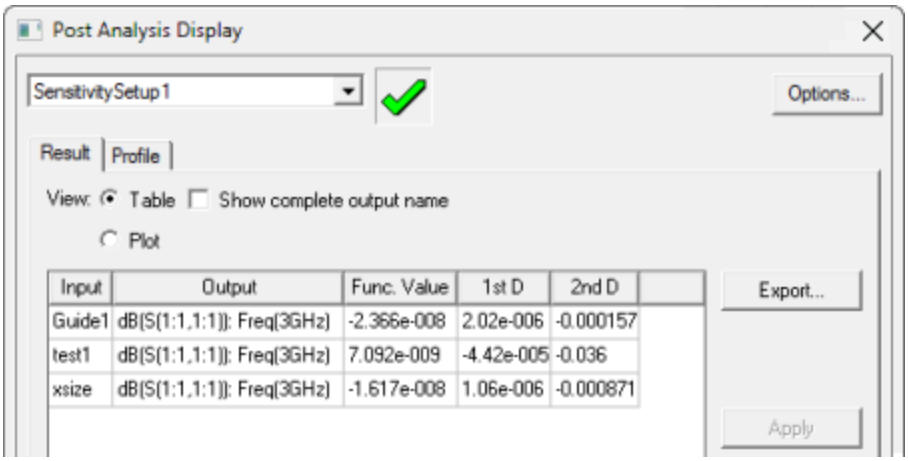
Transpose the table for an alternate view (double-click data table to view the **Properties** window, and on the **Data Table** tab check **Transpose**).



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**.

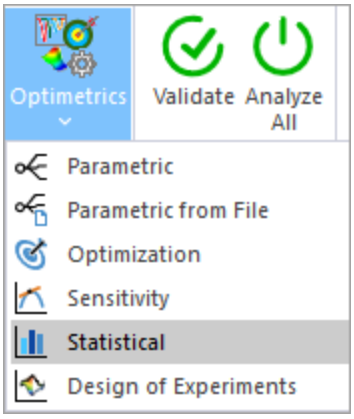
> **Note:**
>
> Statistical Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

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 **HFSS 3D Layout** > **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 HFSS 3D Layout 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 **HFSS 3D Layout** > **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.

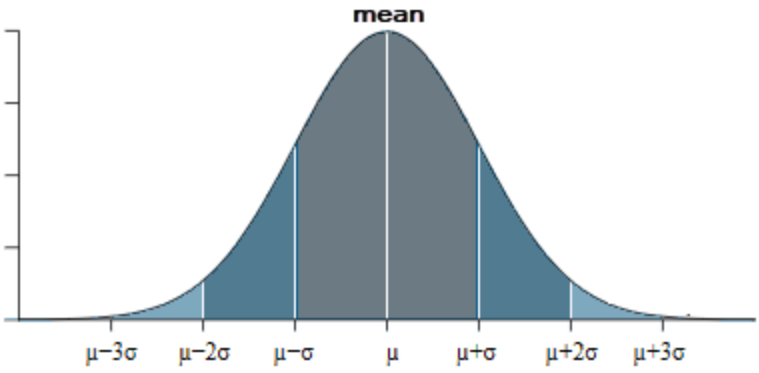
7. If the distribution type is **Uniform**, do the following:
    a. Type a cutoff probability value in the **Cutoff Probability** text box.
    b. Type mean and tolerance values in the corresponding text boxes.

    The design variations are solved within the tolerance range of the starting value, using an even distribution.

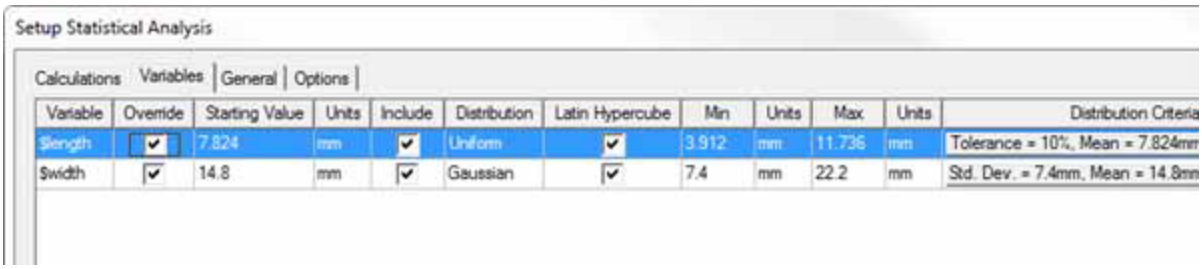
8. If the distribution type is **Lognormal**, do the following:
    a. Type a cutoff probability value in the **Cutoff Probability** text box.
    b. Type values for Sigma, M, and Theta in the corresponding text boxes.
9. If the distribution type is **User Defined**, do the following:
    a. Type a cutoff 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.



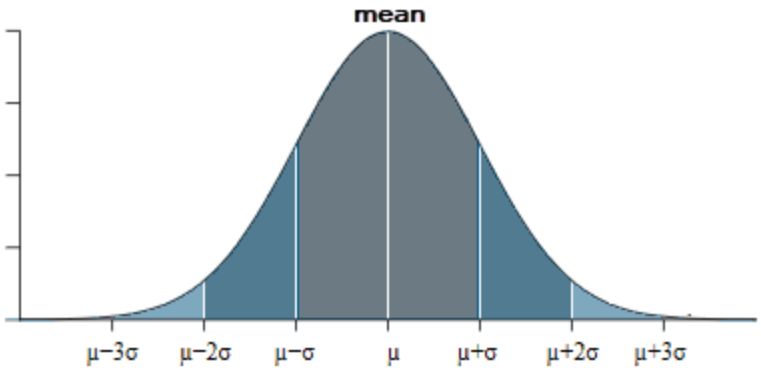
Uniform distributions such as **length** variable above use only the **Tolerance** value and do not have a cutoff probability.

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.

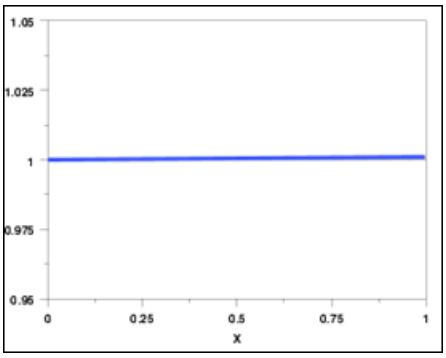
   The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.



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.

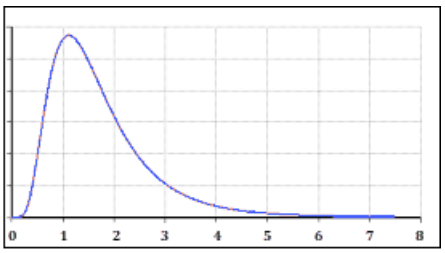
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

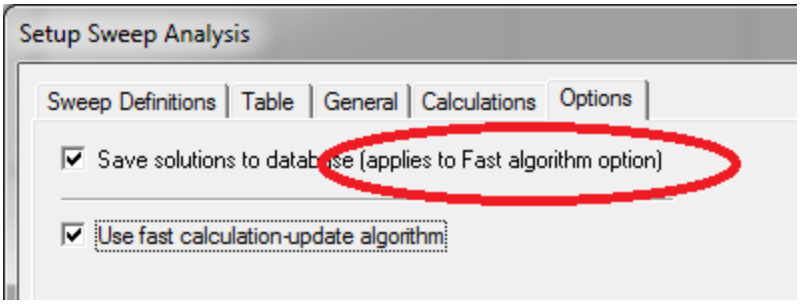
Using the Fast Calculation-Update Algorithm

A fast calculation-update algorithm is available to speed up Optimetrics and report updates during Optimetrics analyses. The fast calculation-update algorithm will generate the same Optimetrics results, only faster, and is available for all Optimetrics analyses *except* a Tuning analysis. By default, the fast calculation-update option is automatically enabled whenever it is applicable, but you can configure it manually using the **Options** tab of the applicable Optimetrics *Setup* dialog box. To enable the fast calculation-update algorithm:

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 your product and then **Circuit** > **Optimetrics Analysis** > **Add** <*Optimetrics Type*>.

   A typical dialog box to *Setup Sweep Analysis* is shown below with the **Options** tab selected.



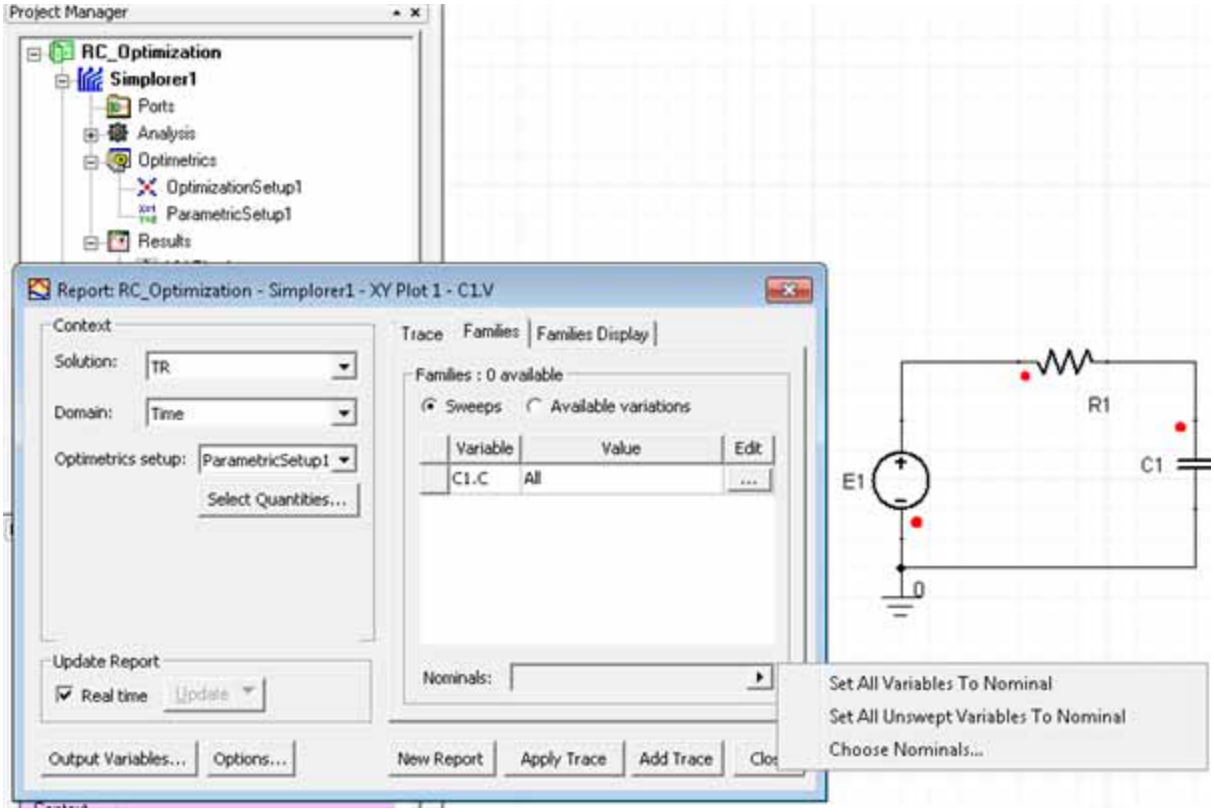
   The *Options* tab is the same for all optimization type setups.

2. Select **Use fast calculation-update algorithm** to enable use of the algorithm. (See also, *Fast Calculation-Update Algorithm Limitations*.)

   When the fast calculation-update algorithm is enabled:

   - You can enable some reports to be updated automatically during the optimetrics analysis if you **Set All Variables To Nominal** in the Report/Trace 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.

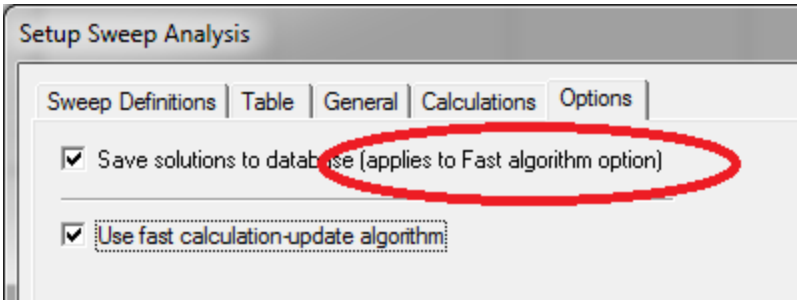


- You can see each trace (overwriting the previous), by setting the **Optimetrics setup** in the Report dialog box to **None** in addition to having all variables set to Nominal. At the end of the analysis, the user will see the last calculated value.

3. If you have enabled **Use fast calculation-update algorithm** and want to save the solution data for every solved design variation in the Optimetrics analysis, select **Save solutions to database** as shown below. Selecting this option has no effect without enabling **Use fast calculation-update algorithm**.

> **Warning:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- When the **Save solutions to database** option is checked, a plot with traces based on the Optimetrics Setup just run can be updated through a menu command. Right-click the desired report under *Results* in the Project Manager and select **Update Report**) to show results as appropriate (including a Family of results, if chosen).
- Without the **Save solutions to database** option checked, you can examine analysis data in the *Post Analysis Display* dialog box, which is available by right-clicking the Optimetrics Setup and choosing **View Analysis Result**.

Fast Calculation-Update Algorithm Limitations

The fast calculation-update algorithm **cannot** be used if **any** optimetrics calculation uses

- **Project/Design variables** – If the project/design variable is ***not*** swept in the Optimetrics analysis, and you would like to use it in the expression: You can create an output variable for the Project/Design variable. Assign the value of the Project/Design variable to the output variable and use the output variable in the expression instead of a constant numerical value.
- **More than one range function** – For example, when range function is not the outermost function or when range function takes arguments.
- **More than one calculation range**, or the calculation range is not for primary sweep.

Similarly, when the fast calculation-update algorithm is enabled, a trace cannot be updated during an analysis if the trace expression uses

- **Project/Design variables**. (However, you can use the same workaround described above.)
- **Any range function.**

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

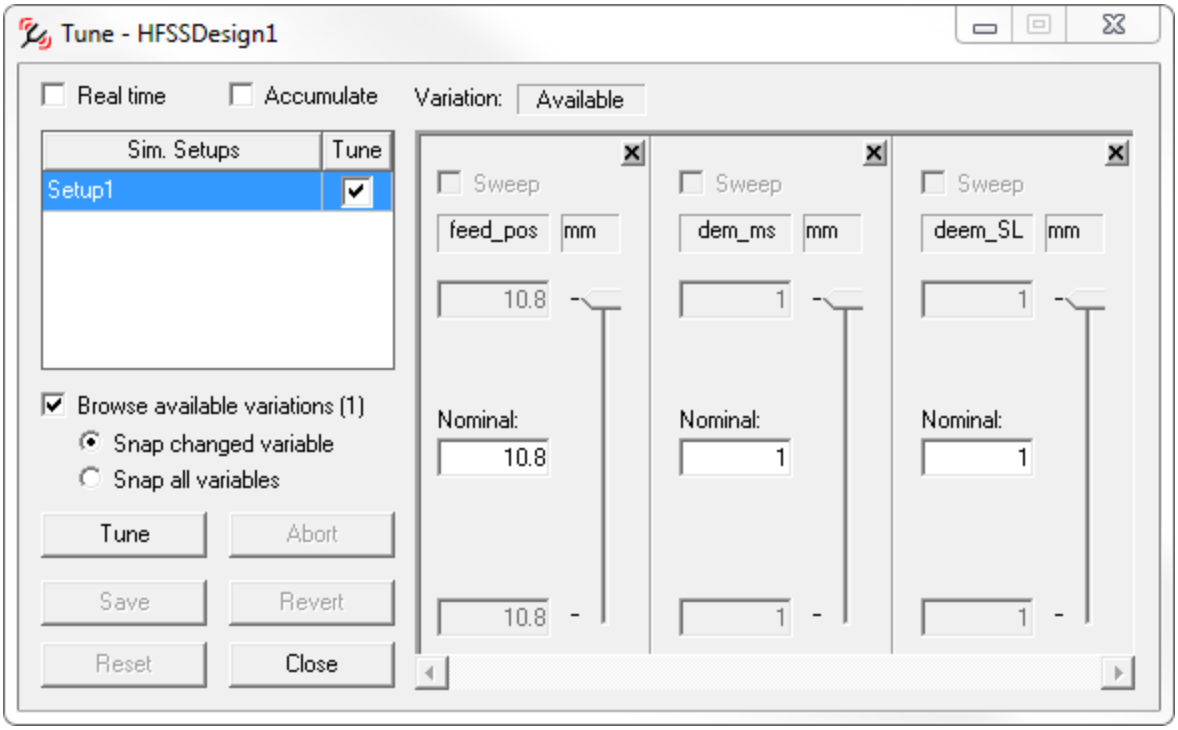
Tuning a Variable

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.

   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.

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 you want to see updates to an open Report plot while tuning a post processing variable, you must select the **Browse available variations** check box. Selecting **Browse available variations** disables the sweep check box, and the fields for minimum and maximum variable values. This feature lets you see the effect of changes to the post processing variables on plotted results.

   Clearing **Browse available variations** enables the Sweep check box, the minimum and maximum fields, and changes the **Nominal** field to **Step**. (See step 6.)

5. In the **Sim. Setups** column, select the solution setup you want to use when it solves the specified design variation.

   The analysis is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

   Checking the Tune box for a Sim Setup enables the Real Time check box, the Browse available variations check box, and the Snap radio buttons. Clearing the Tune box disables those selections.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

8. Changing a variable value with the sliders of by typing in the text field enables the **Save** and **Reset** buttons.

   Clicking **Save** opens a **Save As** dialog box with a name field and an **Apply tuned values to design** check box.

   Clicking **Reset** changes the variable values back to what they were originally.

9. If you have changed one or more included variables, clicking **Close** on the **Tuning** dialog box opens the **Apply Tuned Variation** dialog box. This lists the included variables and the values for each tuning. If you have tried multiple values, they are listed, and the current value is highlighted. Select another value to change the highlight. Click **OK** to apply the highlighted values to the design, or **Don't Apply** to ignore the changes from the original variable values.

   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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

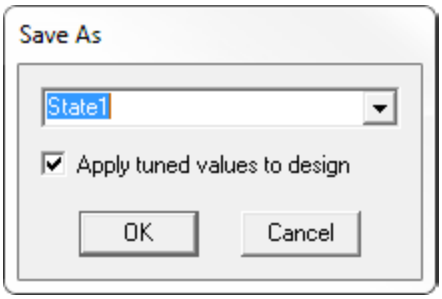
changes affect plots or physical features of a model, those changes should also be apparent.

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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Copying Meshes in Optimetrics Sweeps

An option in the Optimetrics Analysis setups allows you to request Ansys Electronics Desktop to copy a mesh that was calculated for one sweep variation for reuse on a geometrically equivalent sweep variation. For example, with this option selected a sweep on a scan angle would not need to generate meshes for each solution. The option is available on the setups for sweeps on parametrics, optimization, sensitivity, and statistics.

To copy and reuse meshes on geometrically equivalent parametric variations:

1. Define a variable for the kind of Optimetrics sweep you intend to set up.
2. Select Ansys Electronics Desktop and then select the appropriate **Optimetrics** > **Add** command to display the associated **Setup** dialog box.
3. Click the **Options** tab in the setup dialog box.
4. Select **Copy geometrically equivalent meshes**.

   Ansys Electronics Desktop copies the mesh for a particular parametric sweep for reuse on each geometrically equivalent sweep variation.

> **Note:**
>
> This option is available with all Optimetrics setups, and is applied when these analyses generate geometrically equivalent values. However, it is most relevant to parametric sweep, where such equivalences are more likely to occur.

The **Copy geometrically equivalent mesh** option is not recommended for use when the frequency is varying, since meshing is frequency-dependent. You may wish to turn this option off when the first geometrically equivalent variation requires numerous passes after the initial mesh, but the other geometrically equivalent variations require fewer additional passes, so that it is cheaper to start with the initial mesh each time.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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**.

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 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.

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

$$\mathrm{cost}(x): R^n \rightarrow R$$

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

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.

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 **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.

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.

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 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.

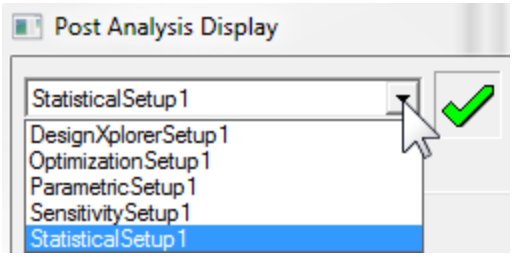
Viewing Analysis Results for Optimetrics Solutions

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.

2. Use the drop-down menu to select from available setups.



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 **Results** tab to view results in plot or table form.

   When you view results in Table format, you can sort the results based on each column. Click a column's header to sort. Click again to invert the current sort.

4. Click **Options...** to open a dialog box that allows you to specify the Maximum number of significant digits to display when showing the analysis result. The default is 4.
5. Select the **Profile** tab to view start, stop, and elapsed times for each variable, and the analysis machine for each variation. Click a column heading to sort the table by variation number, variable value, start, stop, elapsed time, or machine.

See the help topics in this section for more details about viewing Optimetrics analysis results.

Viewing Solution Data for an Optimetrics Design Variation

To view the or computing resources used for any design variation solved during an optimization analysis, you must first select the design variation in the **Set Design Variation** dialog box. This dialog box is accessible from the **Solutions** window and via the **Results** > **Apply Solved Variation** command.

1. Click the **HFSS 3D Layout** 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 HFSS 3D Layout as it solved each design variation. Optimetrics writes the variation information to the profile table before the solve. It then updates the entry with end data (end time, elapsed time, etc.) once the solve variation is completed.

1. In the project tree, right-click the Optimetrics solution setup of interest, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Click the **Profile** tab.
3. Select the Optimetrics setup with the results you want to view from the drop-down menu at the top of the dialog box.

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 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.

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:

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, 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**.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

   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.

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 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.
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.

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 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

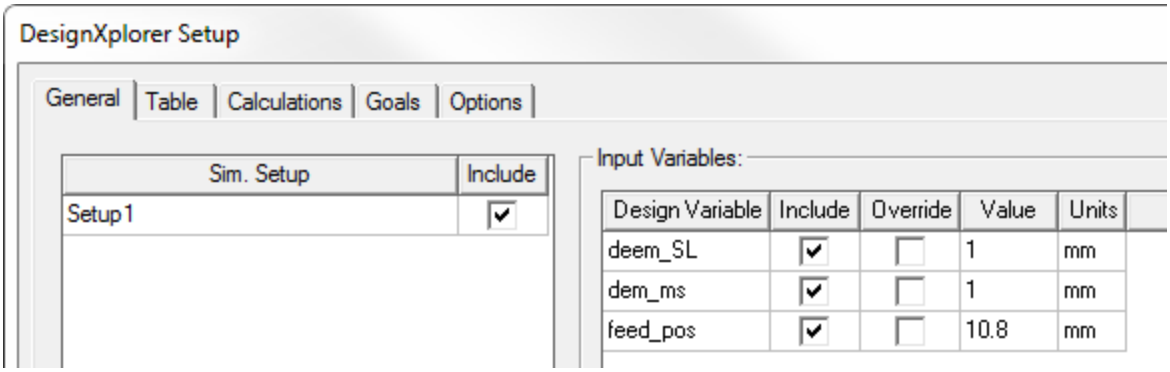
You can export a .xml file containing information on an HFSS 3D Layout setup, optimization variables, and output variables that enables Ansys Design Xplorer to manage the simulations (for example, for design of experiments and optimization). Design Xplorer will launch Ansys Electronics Desktop simulations of design variations and evaluate the outputs.

To do so:

1. Click your product on the menu bar and then **Optimetrics Analysis** > **Add Design Xplorer Setup** or right-click on **Optimetrics** in the **Project Manager** window, and select **Add Design Xplorer Setup** from the short-cut menu.

   This opens the **Design Xplorer** dialog box with the **General** tab selected. It lists the setups available in the current project, and the input variables it contains.

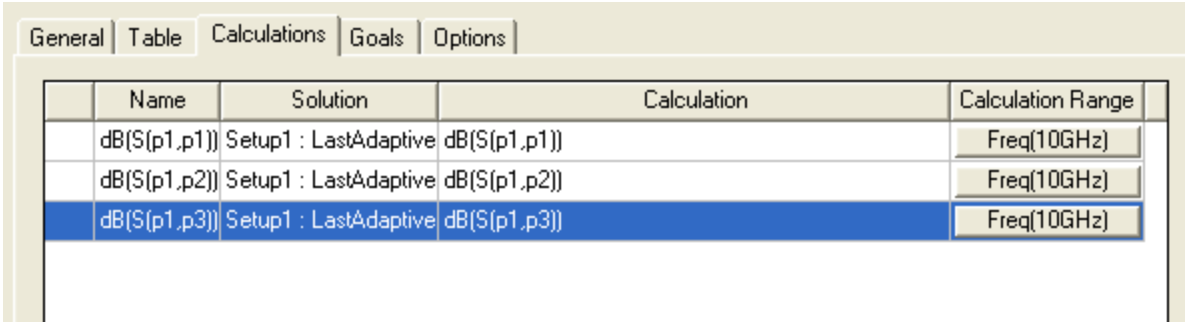
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select **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**.

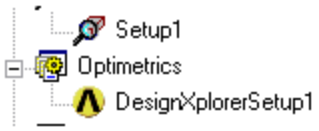
   This opens the **Add/Edit Calculation** dialog box. Here you can define the simulation results of interest. The dialog box contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**. See: Setup Calculations for Optimetrics for details.

   Use the **Add Calculation** button to add expressions to the Calculations table.



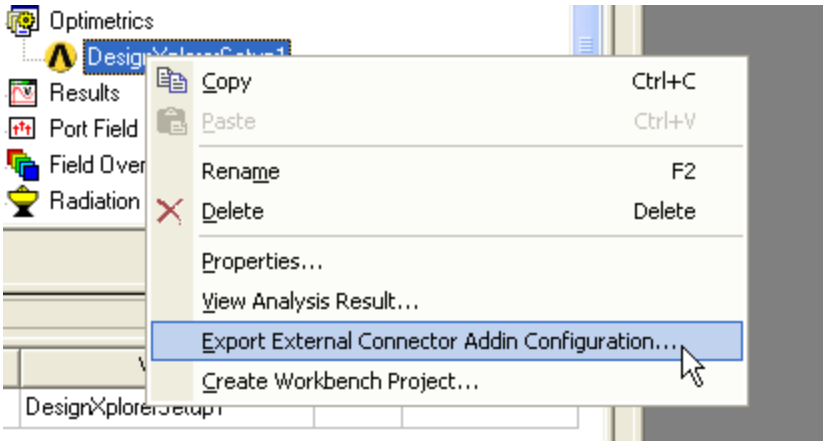
5. When you have added the calculations of interest, click **OK** to save the setup.

   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.

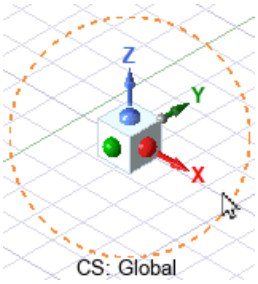
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13 - Modifying the 3D Model View

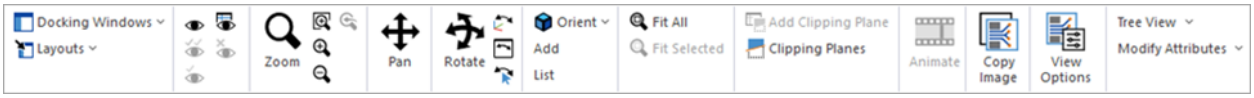
You can modify the view of objects in the 3D **Modeler** window without changing their actual rotation or position within the coordinate system. You vary the viewing direction and the relative location of the "camera" (that is, the viewpoint) relative to the model geometry. The tools allow you to zoom, pan, or rotate the view orientation and to quickly switch to predefined or user-defined view orientations. Additionally, you can choose different object visualization options, show/hide objects, and customize the window background.

What do you want to do?

Change the View	Show or Hide Design Objects
Change Object Visualization	Change the Background



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

- Changing the Model View with Alt+Double-Click Areas in the Modeler Window
- Apply an Orientation to the Current View
- Set the View Options
- Rotate the View
- Pan the View
- Zoom In or Out

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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

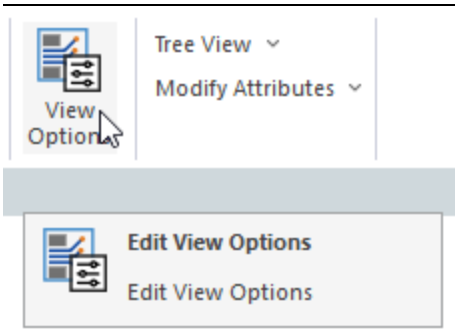
Change the Background

- Setting the Background Color
- Modify the appearance of the Coordinate System Axes
- Modify the appearance of the Grid

View Options: 3D UI Options

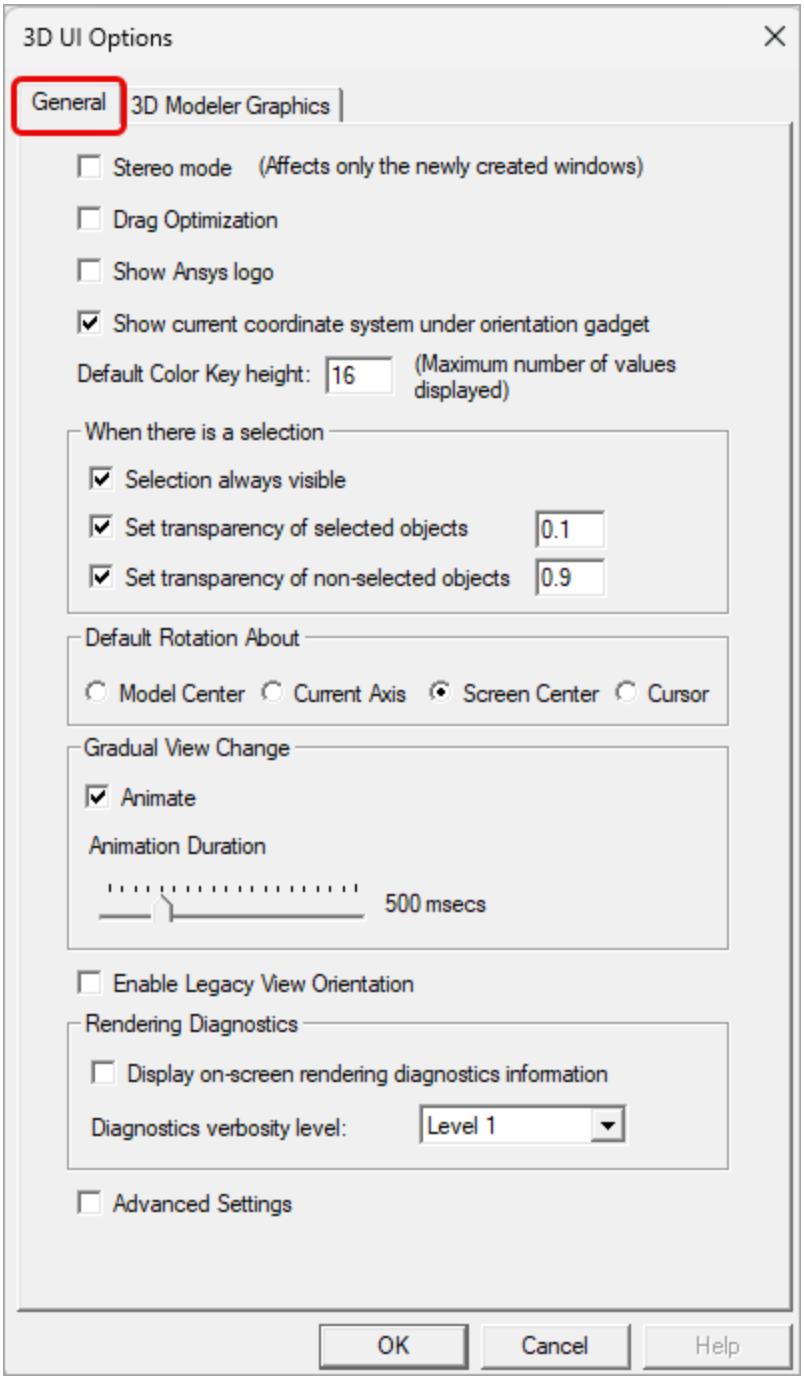
To access View Options, use the **View** > **Options** command, or click **View Options** on the **View** ribbon.

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

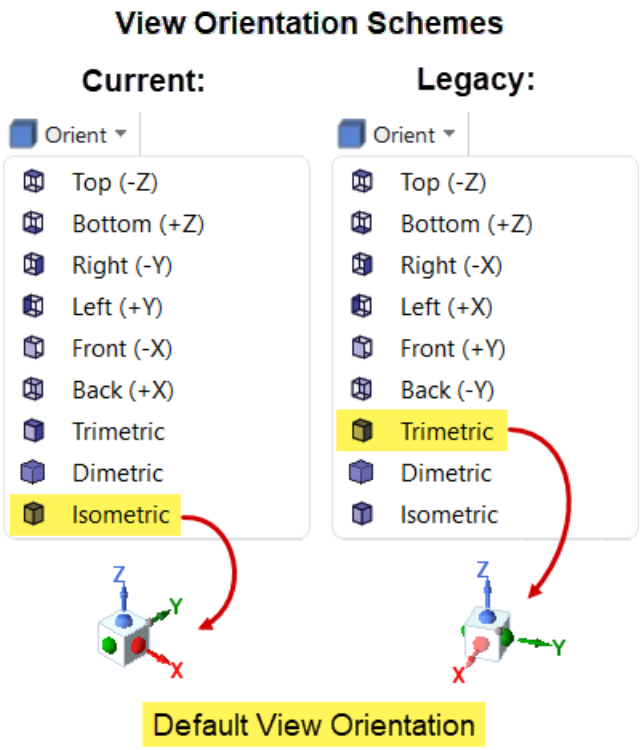
  For operations such as Fit Selection, Fit All, and Zoom, these settings determine whether Electronics Desktop animates the transition between images and how smoothly it does so.

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:



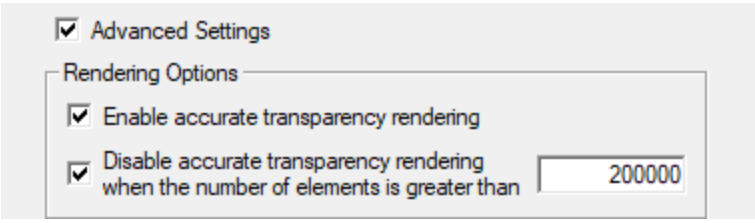
Rendering Diagnostics Options: .

You can select to display on-screen rendering information, and the diagnostics verbosity level from level 1 to level 4.

Advanced Settings: Rendering Options:

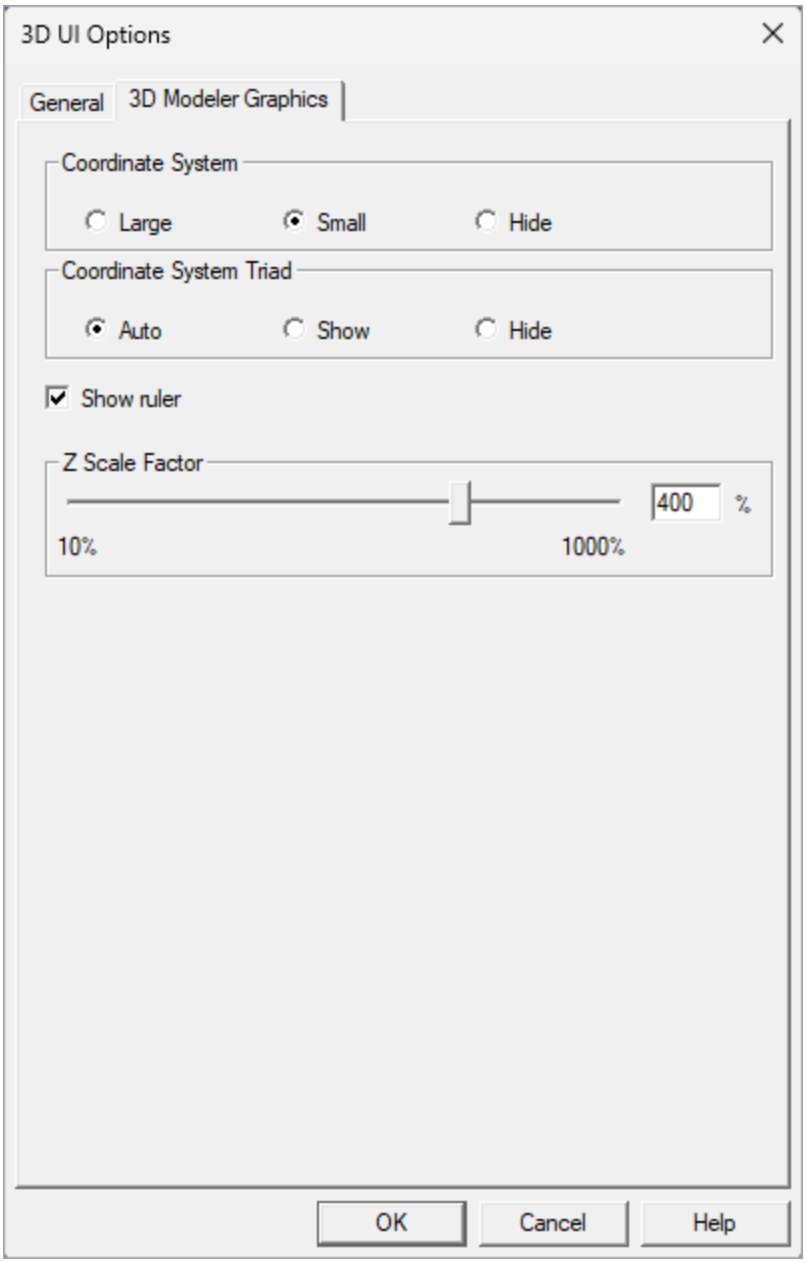
Because rendering transparency correctly can be compute intensive, especially for complex models with thousands of objects, it can bring down the frame rate. In such cases, you can select whether to **Enable accurate transparency rendering**.

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

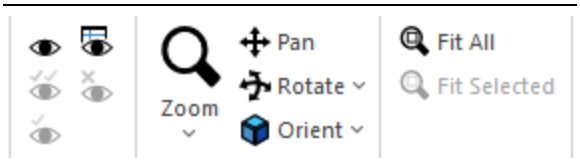
To move (pan) the view:

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** + "**-**"

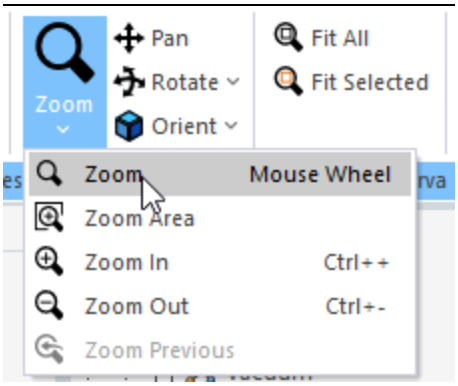
  The view zooms out 5 percent.

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**.

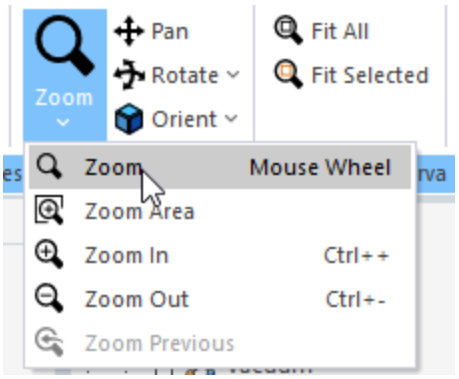
Zoom In and Zoom Out Commands

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.

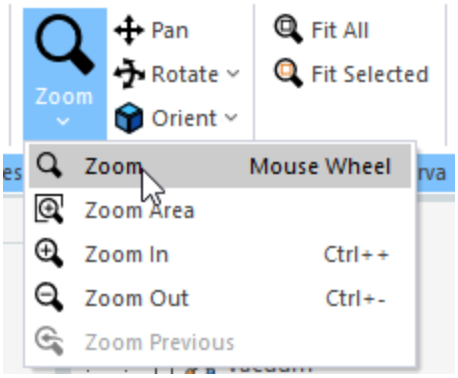
   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

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**.

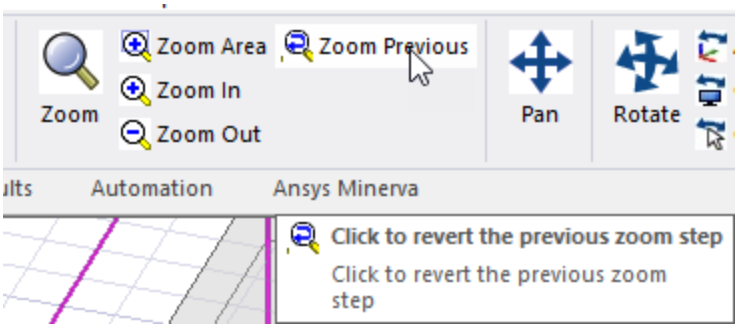
Zoom Previous Command

The **Zoom Previous** command is not enabled until at least one **Zoom** command has been used.



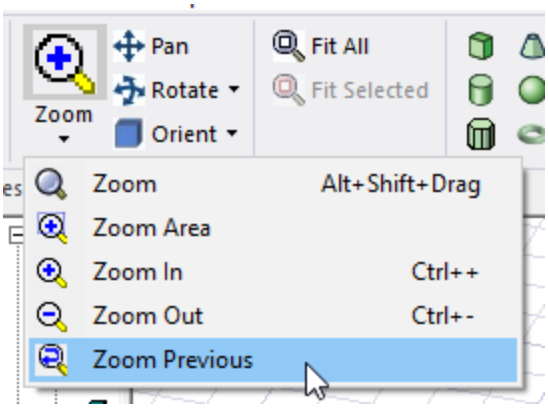
It operates on any existing sequence of **Zoom** commands. That is, it undoes the last previous **Zoom** command, and then, another exists, the one before that. To use **Zoom Previous**.

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:

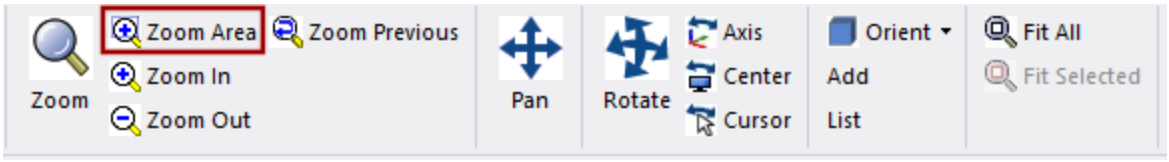
If zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**

For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

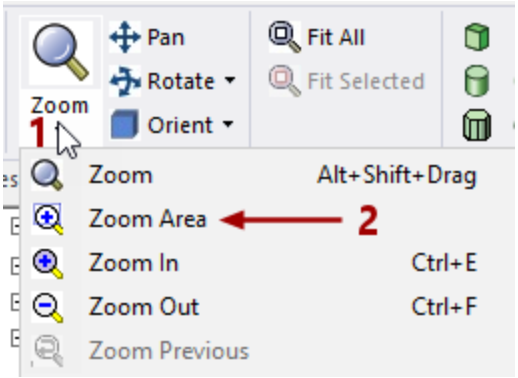
Zooming into a Rectangular Area

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**.

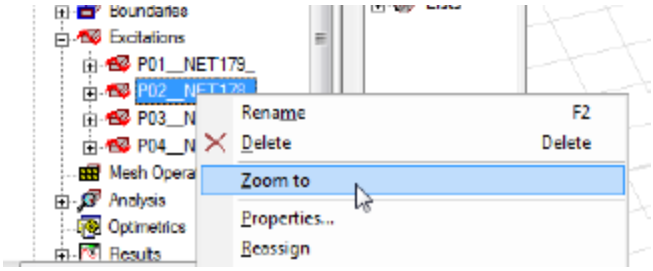
Zoom to a Selected Boundary or Excitation

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.

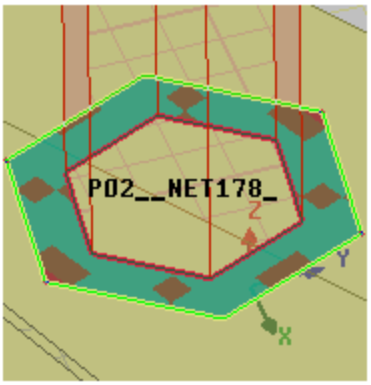
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

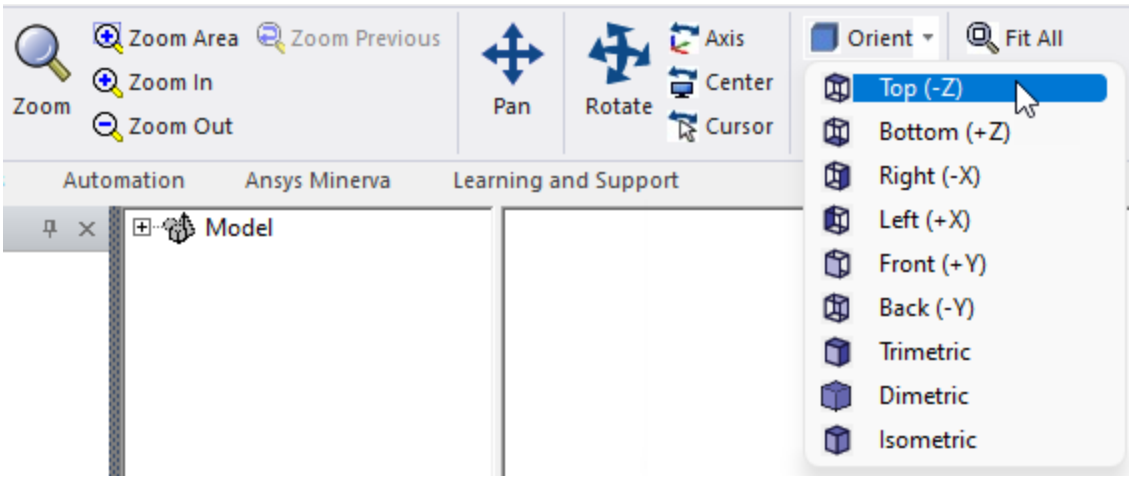


Modifying the View Orientation

There are several places you can access commands to change the view orientation in a *Modeler* or *3D Plot* window:

- Select the **Orient** drop-down Menu on the **View** ribbon tab and choose one of the predefined view orientations:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

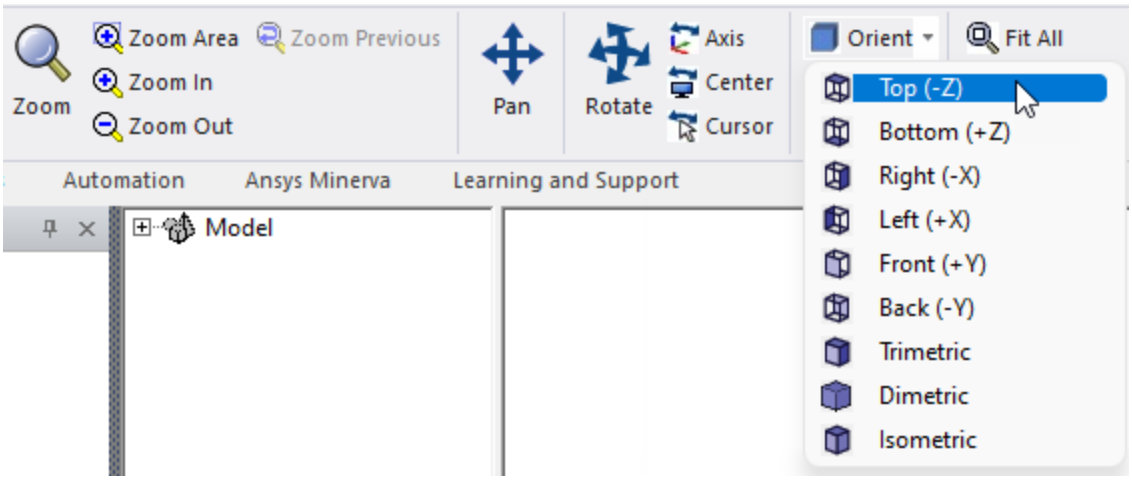


The sign and letter in parentheses that follows each standard view indicates the viewing direction, not the model side. For example, consider a simple box aligned with the global planes. The right side of the box is the +X side, but the viewing direction is -X (you look leftward to see the right side of an object).

Apply an Orientation to the Current View

Apply view orientation settings using any one of several available methods:

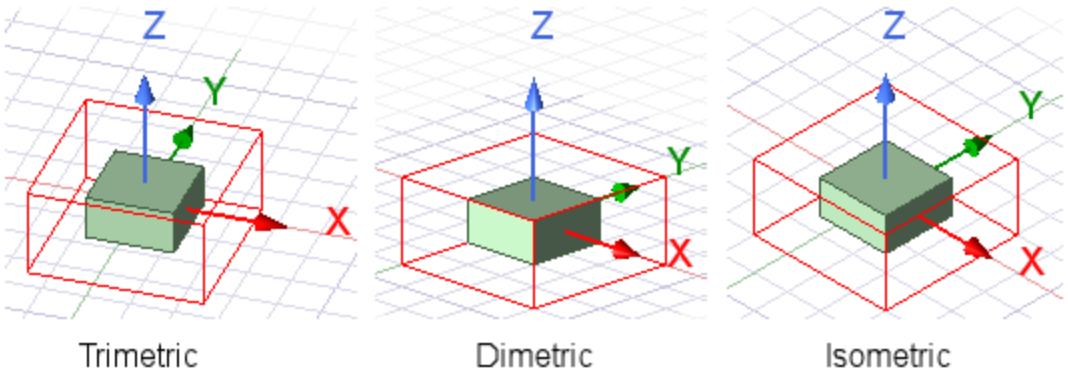
- Use the commands in the **Orient** drop-down menu located on the **View** ribbon tab:



Possible orientations are Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Isometric. These last three differ as shown below:



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.

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):**

  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.

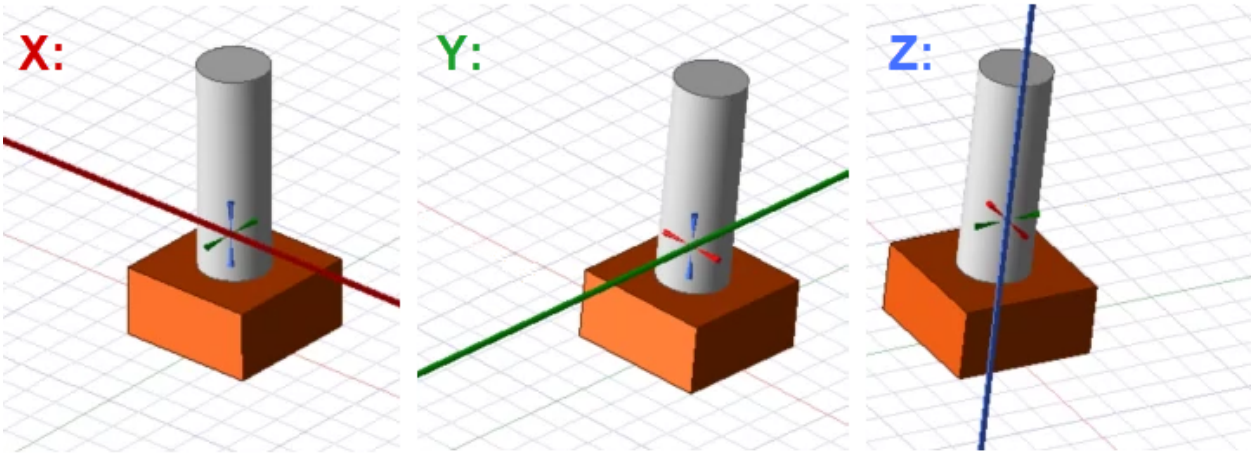
- **Constrained rotation about the X, Y, or Z axis**:

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 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:



Note:

If rotation response is slow, especially for complex models:

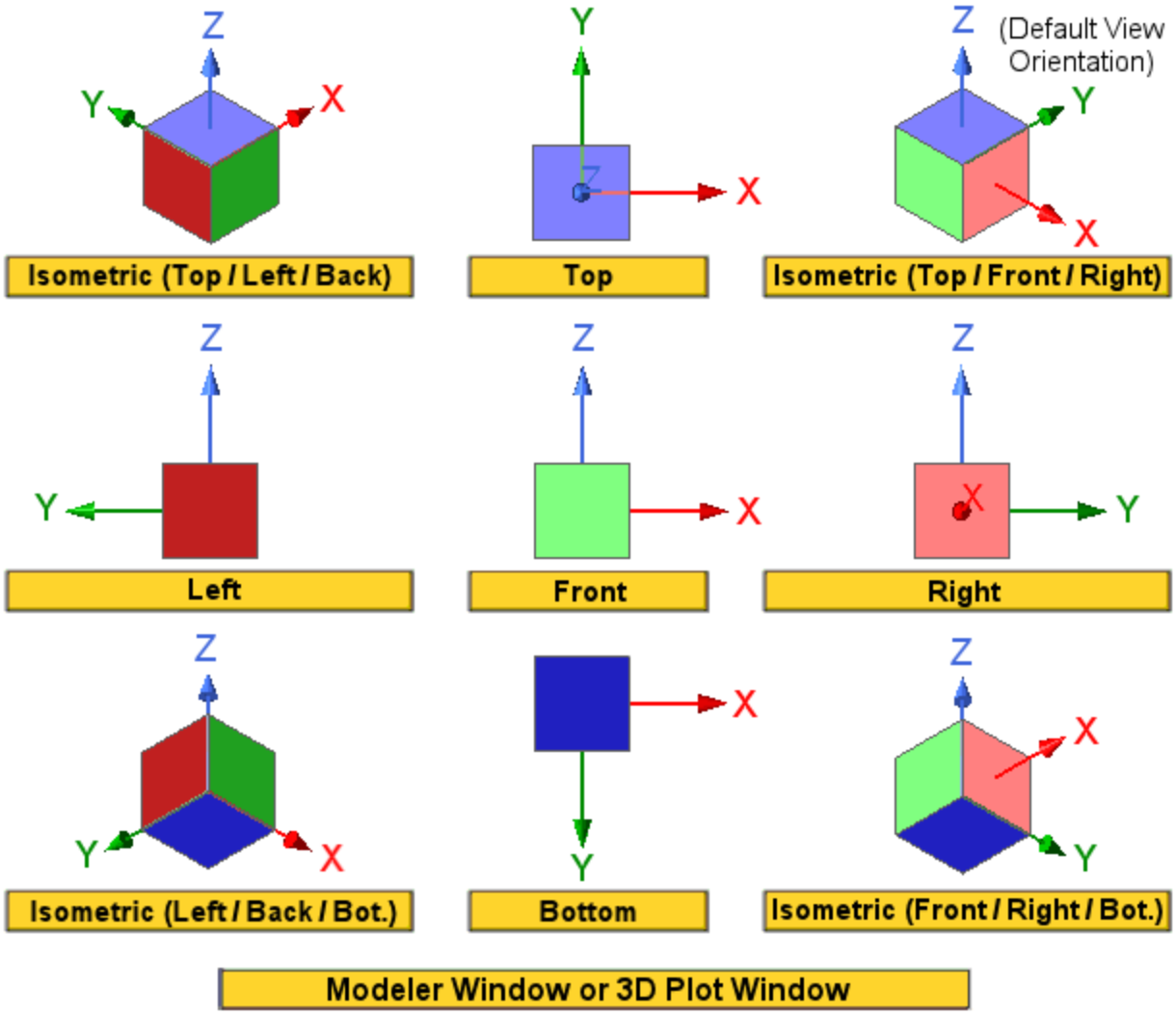
- For some graphics cards, you can improve the performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**.

  For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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).



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.

Fitting Objects in the View Window

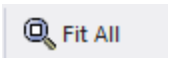
What do you want to do?

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Fit all objects or traces in a view window
- Fit selected objects in a view window

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.

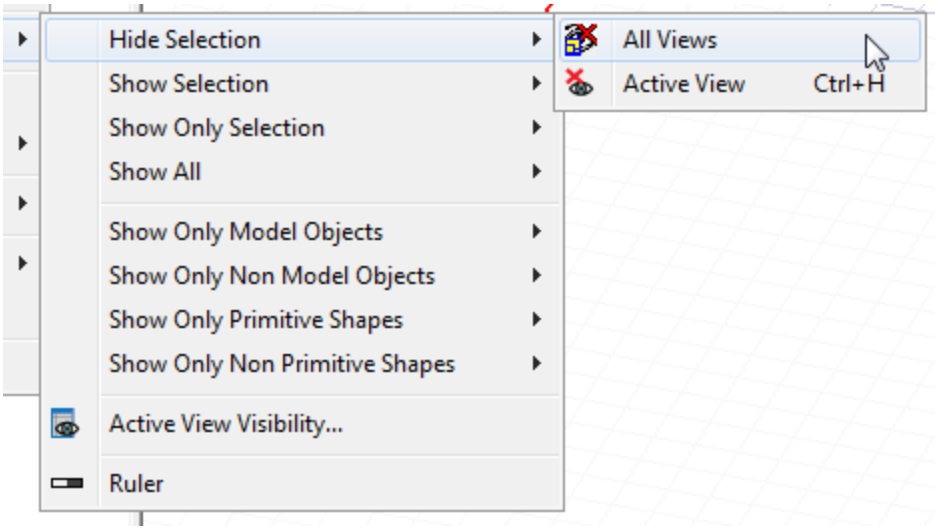
Hiding Objects from View

To hide selected objects.

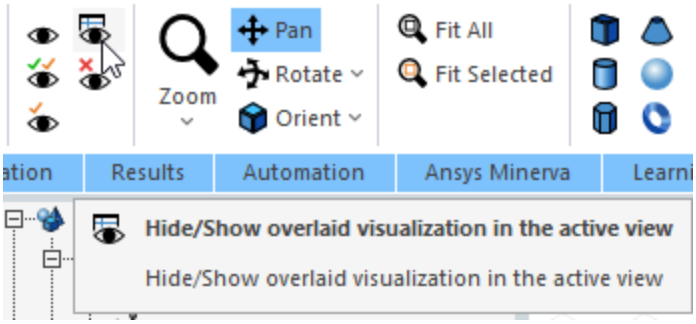
1. Select the object you want to hide from view. This enables the menu commands and **View, Draw,** and **Model** ribbon tabs icons for hiding objects.
2. Click **View** > **Visibility** > **Hide Selection** and one of the following commands:
   - **All Views** to hide the selected object in every open 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.

   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**.

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 **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.

> **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.

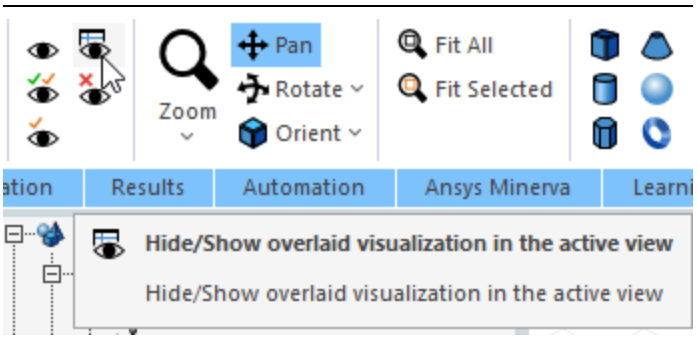
Object visibility is saved with the project.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

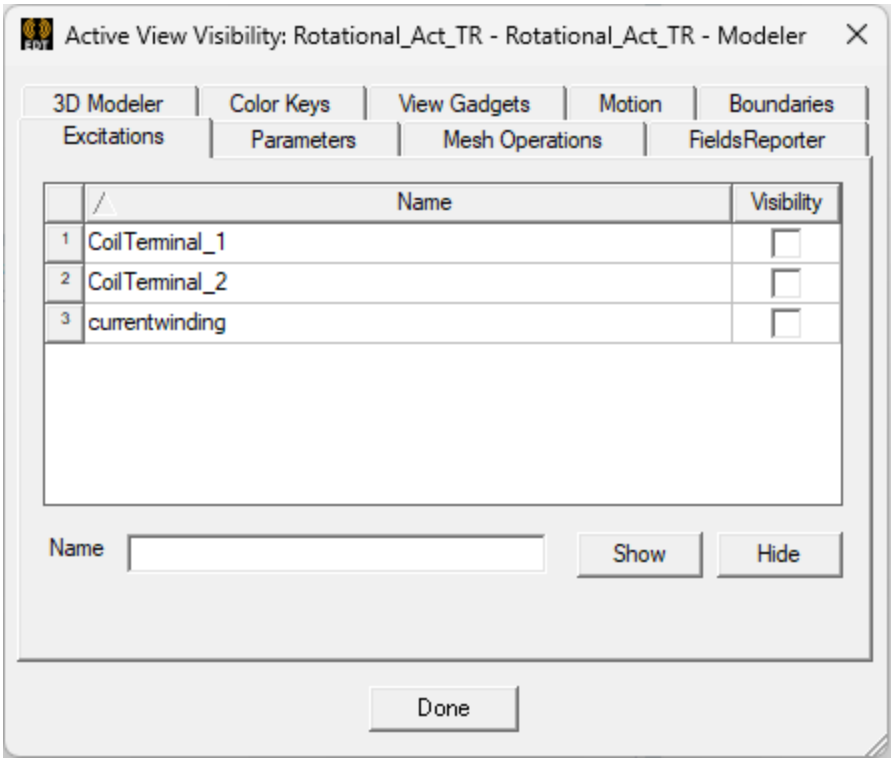
Active View Visibility Dialog Box

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.

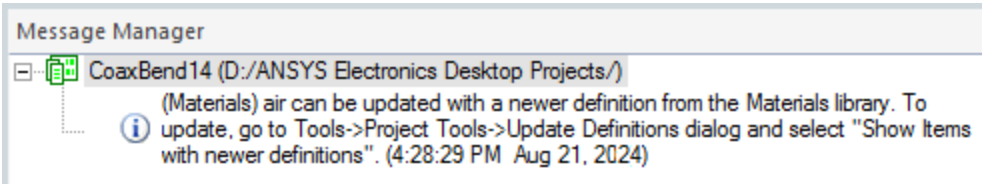
To set the Enhanced Display for the active modeler window:

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.

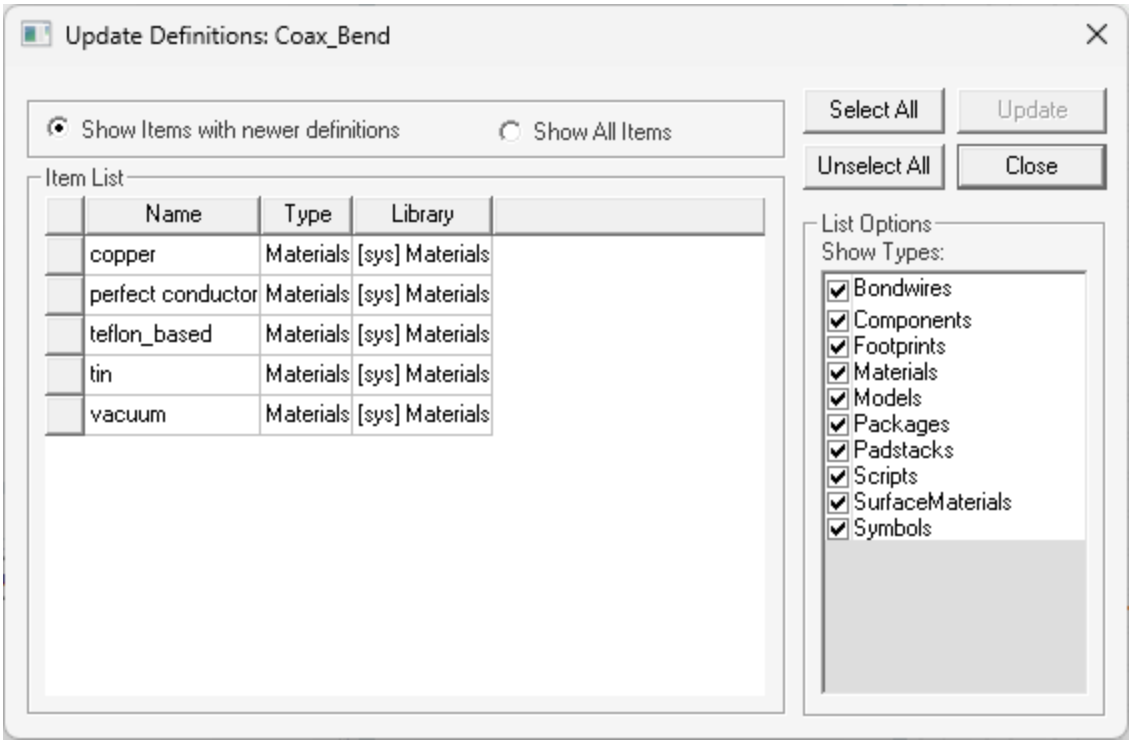
It is possible for the view to be in the “Enhanced Display” mode but some of the parts in the model could have “Use Material Appearance” unchecked. In this case you can edit the **Color** or **Transparent** values, or select **Use Material Appearance** in the docked **Properties** window, but they are not applied to the active view visualization. This resembles how the **Wire Frame** setting is handled currently.

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.



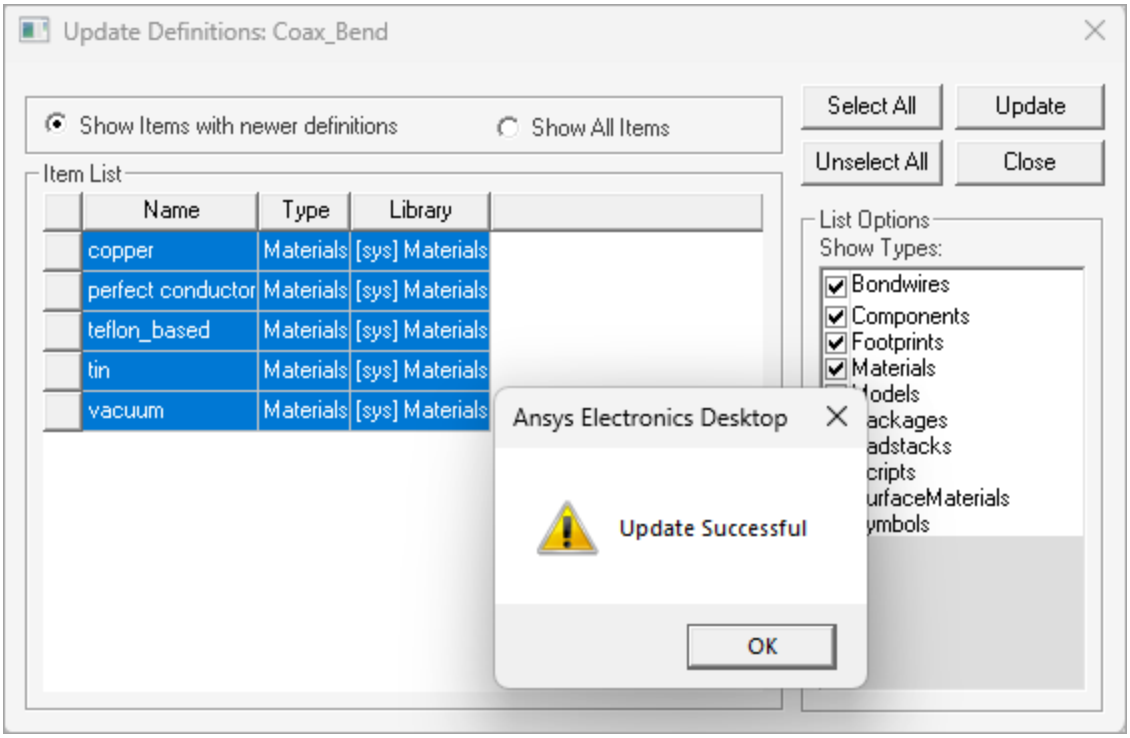
If you want to update, click **Tools** > **Project Tools** > **Update Definitions** to open the **Update Definitions** dialog box.

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 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.

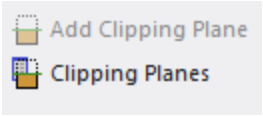
Using Clip Planes

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.

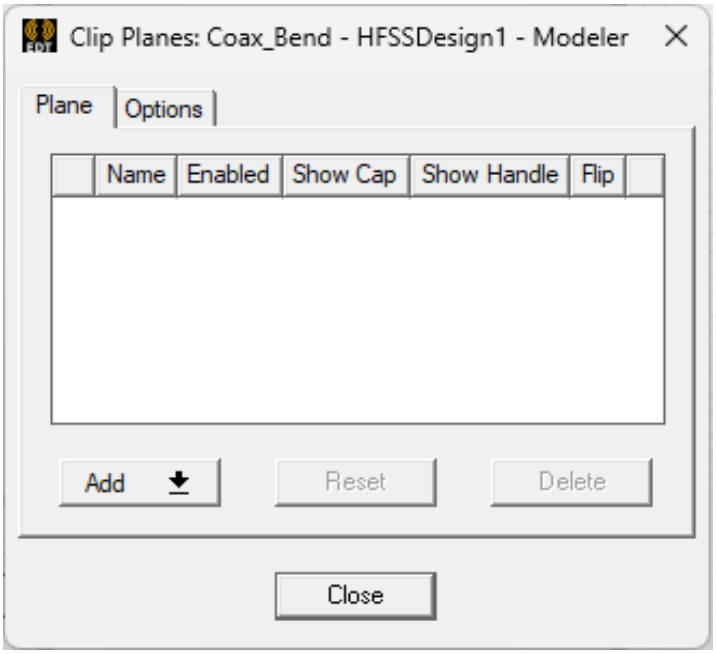
To add a clip plane:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

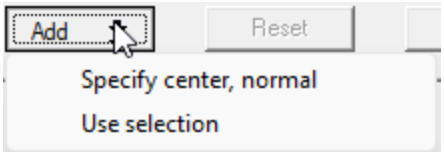
1. Click **View** > **Clipping** > **Clip Plane** or click the **Clipping Planes** icon on the **View** ribbon tab.



This displays the **Clip Plane** dialog box with the **Plane** tab selected.

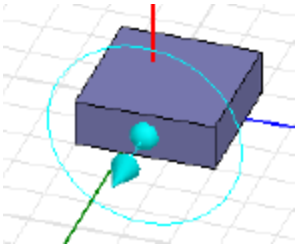


2. The **Add** button contains a drop-down menu with choices for **Specify center, normal**, and **Use selection**. If you want to use a selection, you must first select a face or an existing cut plane.

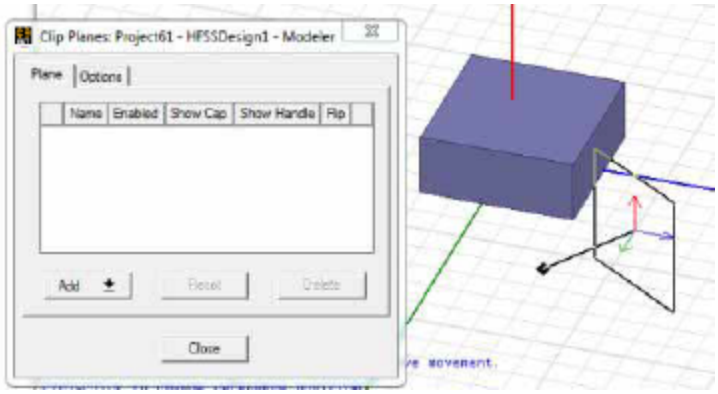


- If you first select a face or cut plane, and then click **Add** > **Use Selection**, the clip plane is added on that face.

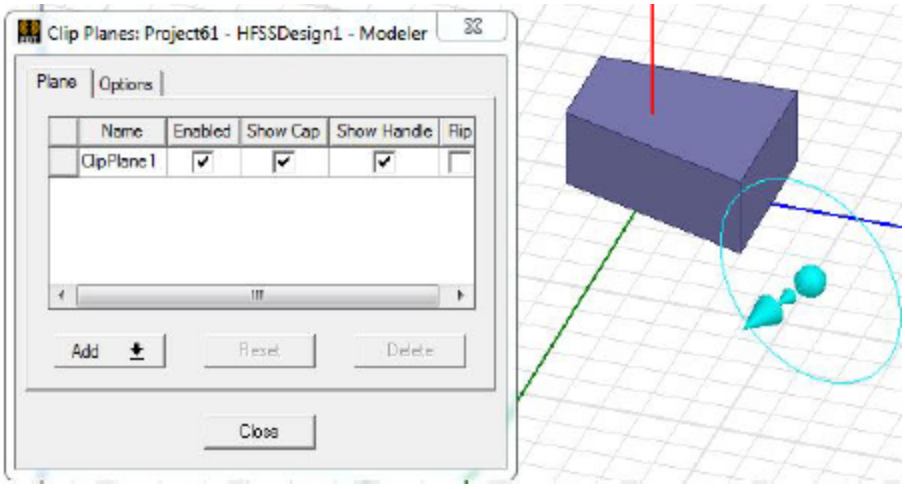
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- 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.



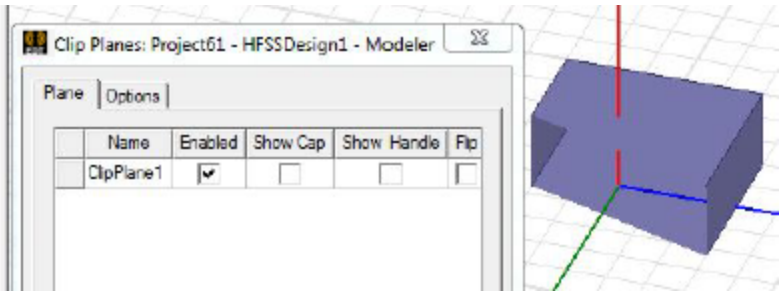
  b. Click again to set the reference position.



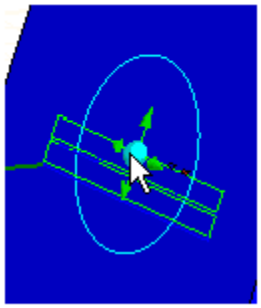
  After the second click, the clip plane is active. The handle is visible as a circle with a sphere at the center, and an arrow pointing the normal for the plane.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

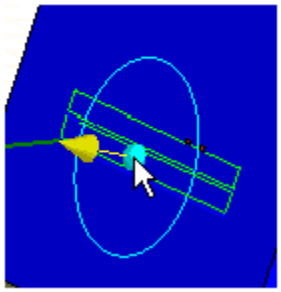
The **Clip Planes** dialog box shows the clip plane name, that it is enabled, the cap (which is the plane surface), and the handle. **Flip selection** lets you reverse the direction of the clip plane. If you deselect **Show cap** and/or **show handle**, they disappear from the display.



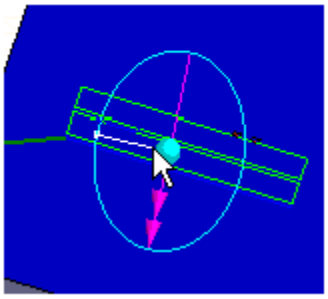
3. With **Show Handle** enabled, you can use the handle to manipulate the location and orientation of the clip plane. The handle changes appearance and function relative to the position of the cursor. Dragging the cursor makes use of the current function.



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

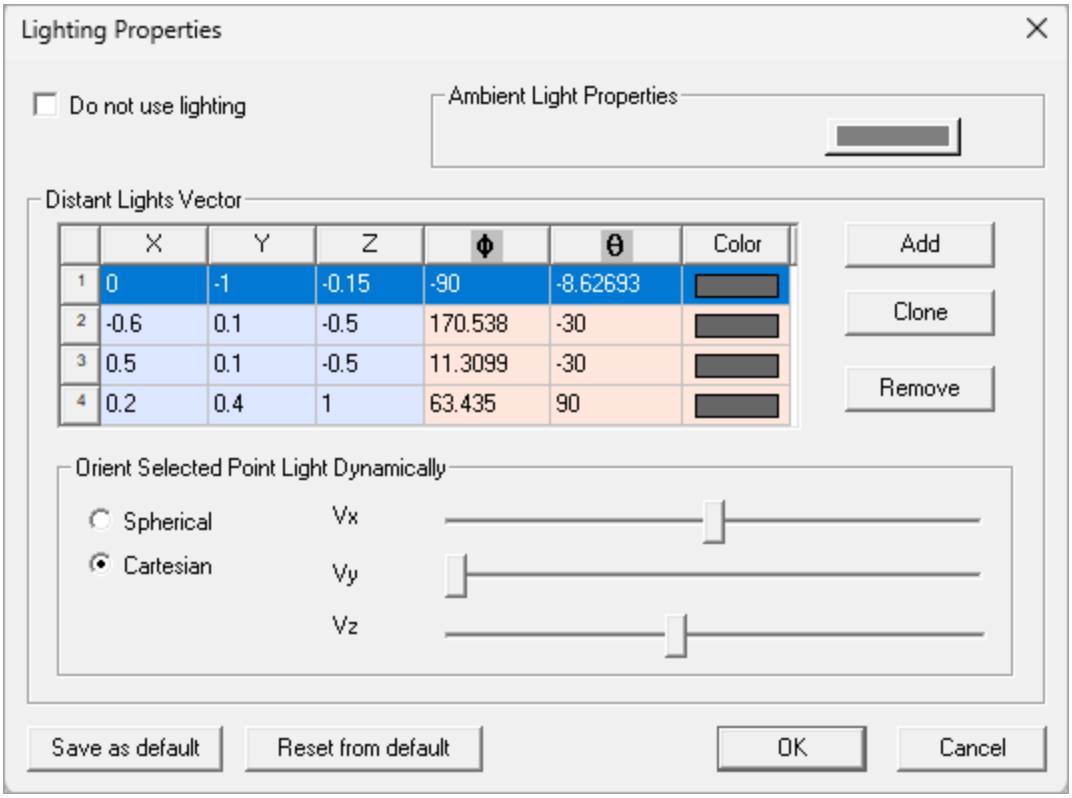
You have the option to emit the following types of light on a design:

- *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.

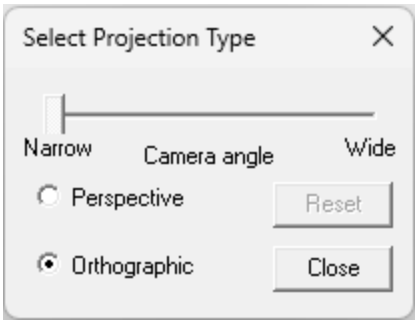
Setting the Projection View

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.

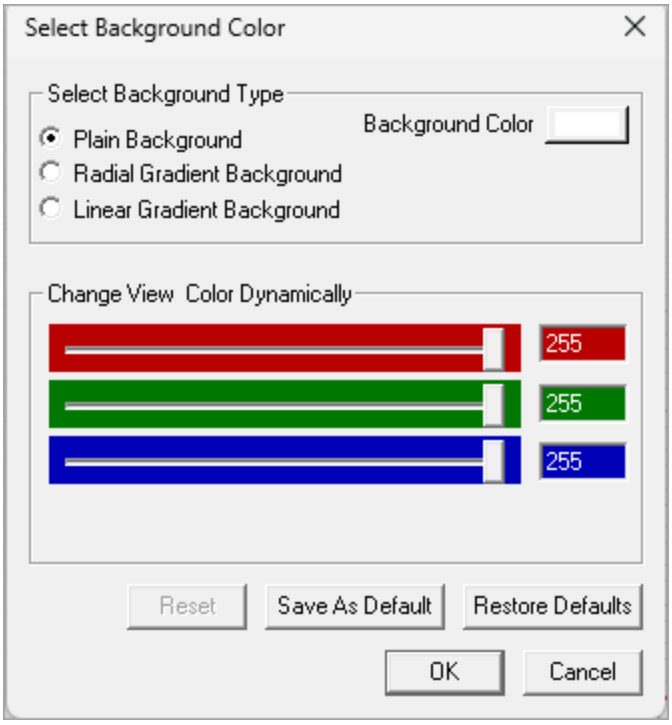
Setting the Background Color

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.

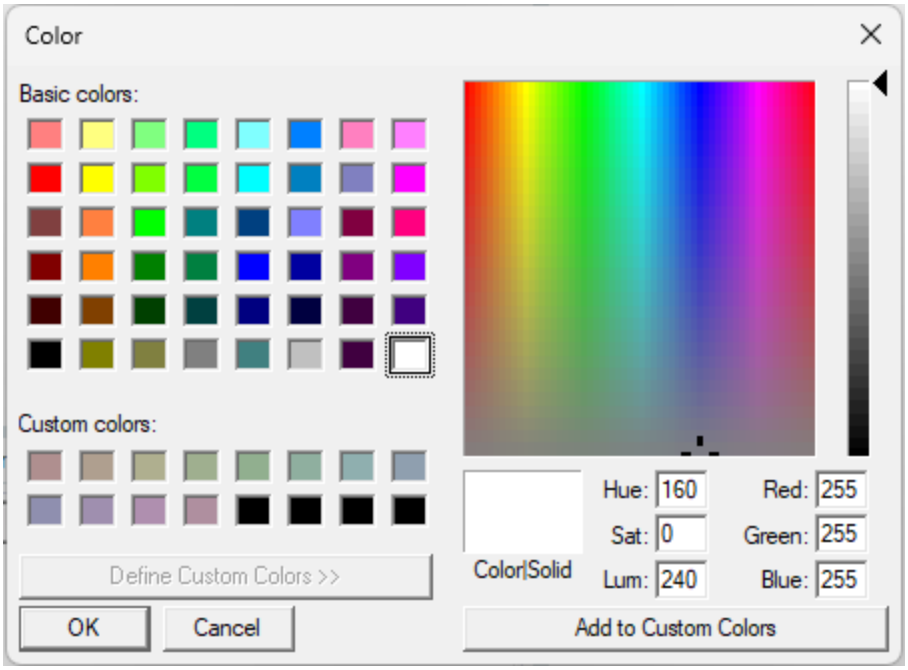
   The **Select Background Color** window appears.

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.

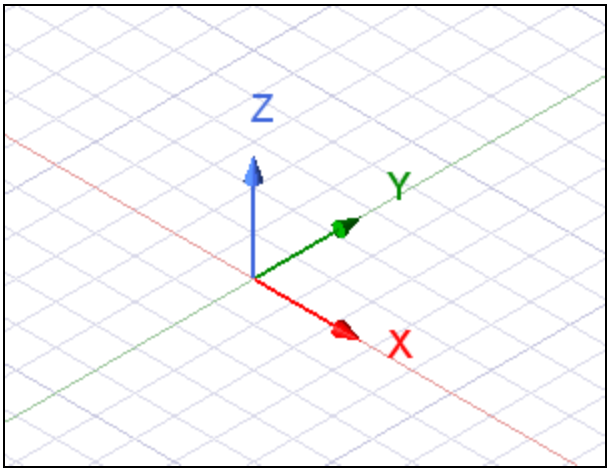
3. To assign a background color that gradually changes from one color to another, do the following:
    a. Select **Gradient Background**.
    b. Specify the background color at the top and bottom of the view window in one of the following ways:
        - Under **Select Background Type**, click the **Top 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:

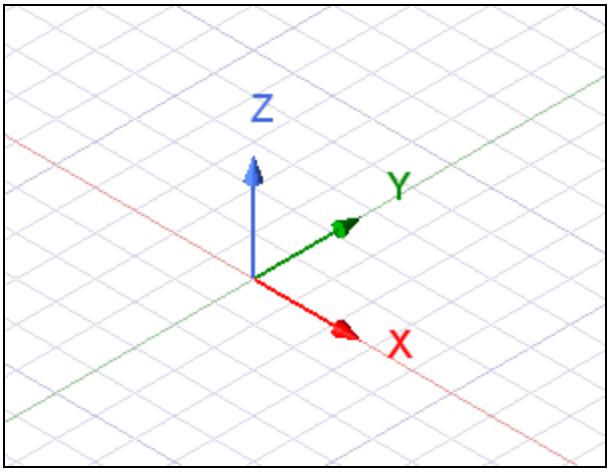
1. Click **View** > **Coordinate System**> **Triad**, then click one of the following:
    - **Hide** to hide the triad x-, y-, and z-axes at the lower right of the active view window.
    - **Show** to display the triad x-, y-, and z-axes in the lower right active view window.
    - **Auto** to generally hide the triad axes.

Choosing Grid Settings

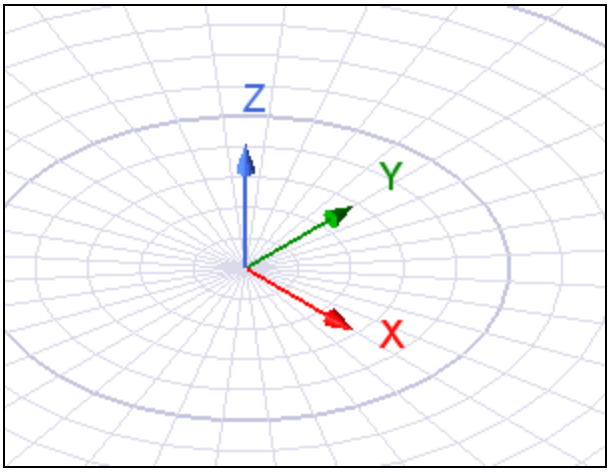
The grid displayed in the **Modeler** window is a drawing aid that helps to visualize the location of objects.

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

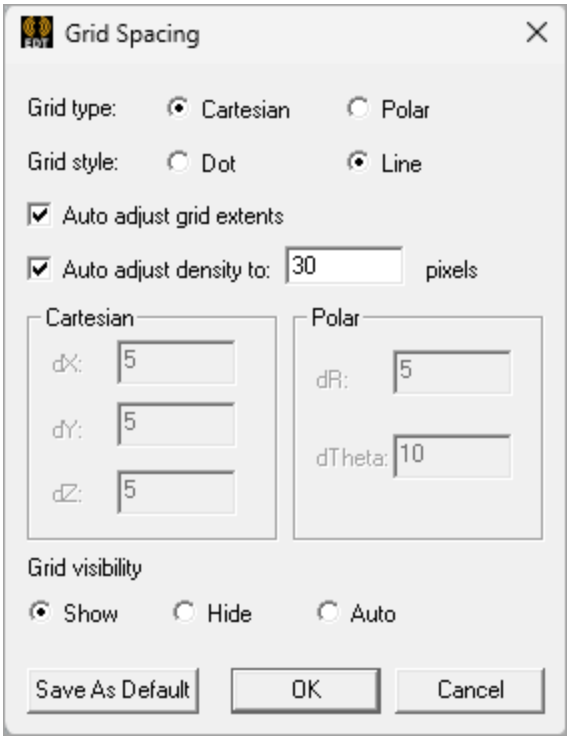
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

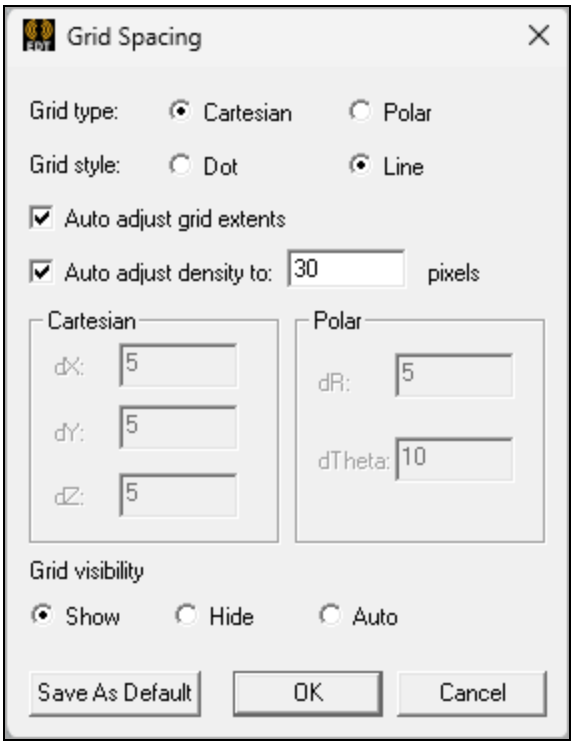
For polar grids, you define a coordinate by specifying its radius from the origin in the **R** text box and its angle from the x-axis in the **Theta** text box or its relative distance from the previously selected point in the **dR** and **dTheta** text boxes.

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.

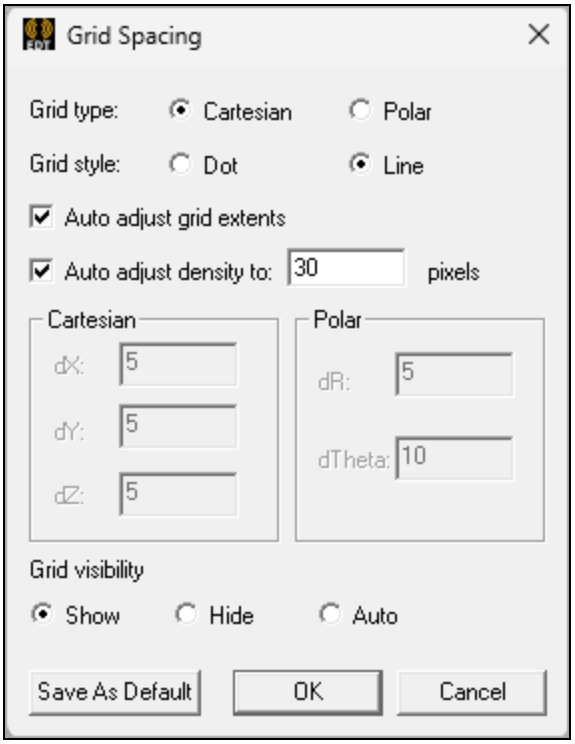
3. To show a Minimal grid around existing objects, check **Auto adjust grid extends**. To display a grid as a plane, deselect **Auto adjust grid extends**.

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.

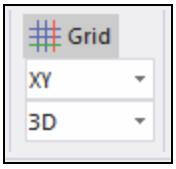
    The distance between grid points will increase and decrease proportionately as you zoom in and out in the active Modeler window.

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

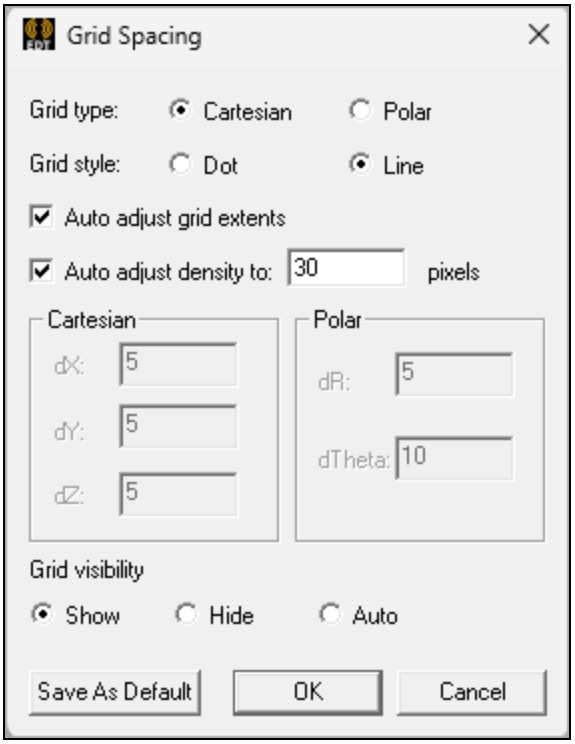
To hide the grid in the Modeler window, on the **Draw** ribbon tab, click the Grid toolbar icon. Click it again to show the grid.



Alternatively:

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.



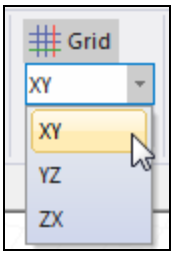
2. Select **Show** to make the grid always visible in the active **Modeler** window. Select **Hide** to make the grid always invisible in the active **Modeler** window. Select **Auto** so the grid appears only while you are drawing an object in the **Modeler** window.

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:



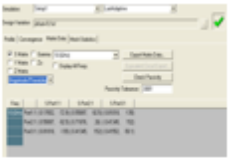
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

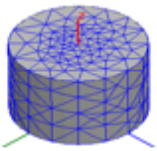
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

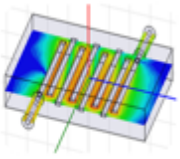
14 - Postprocessing and Generating Reports

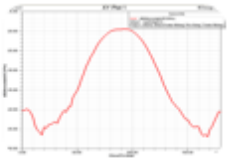
As Ansys Electronics Desktop completes a solution, you can display and analyze the results:

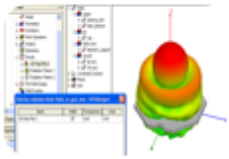
View Solution Data

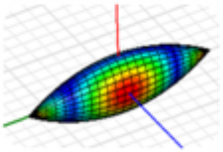

View Optimetrics Results

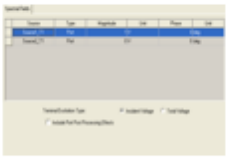

Create Mesh Plots

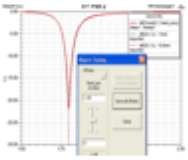

Plot Field Overlays

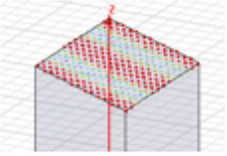

Create or Modify 2D or 3D Reports


Overlay 3D Field Plot on Model


Create Animations

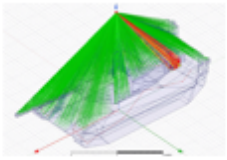

Scale Excitation Mag/Phase

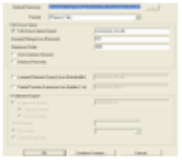

Tune Reports


Port Field Display

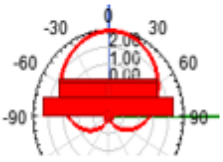
Specify Output Variables

Radiated Fields Observation Regions

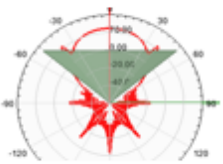

Visual Ray Trace

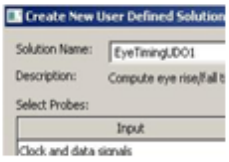

Export Equivalent Circuit Data

Fields Calculator / Derived Field Quantities



Overlay 2D Radiation Fields


Overlay Monostatic/ Bistatic RCS



User Defined Outputs

User Defined Documents

Network Data Explorer

- View solution data including the following: convergence information, computing resources that were used during the solution process, mesh statistics, and matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution. For eigenmode solutions, you can view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode. Solution data can also be viewed while Ansys Electronics Desktop is generating a solution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- View analysis results for Optimetrics solutions.
- Plot field overlays - representations of basic or derived field quantities - on surfaces or objects.
- Overlay Field Plots on Models
- Create 2D or 3D reports of S-parameters, basic and derived field quantities, and radiated field data.
- Plot the finite element mesh on surfaces or within 3D objects.
- Create animations of field quantities, the finite element mesh, and defined project variables.
- Scale an excitation's magnitude and modify its phase.
- Apply Derivative Tuning to Reports

> **Note:**
>
> Except in the case of non-model boxes drawn in the global coordinate system (CS), non-model objects cannot be used for any fields post processing operation You can use non-model boxes drawn in the global CS for post processing operations, including integration and solution domaining.

Fields Post Processing in HFSS 3D Layout

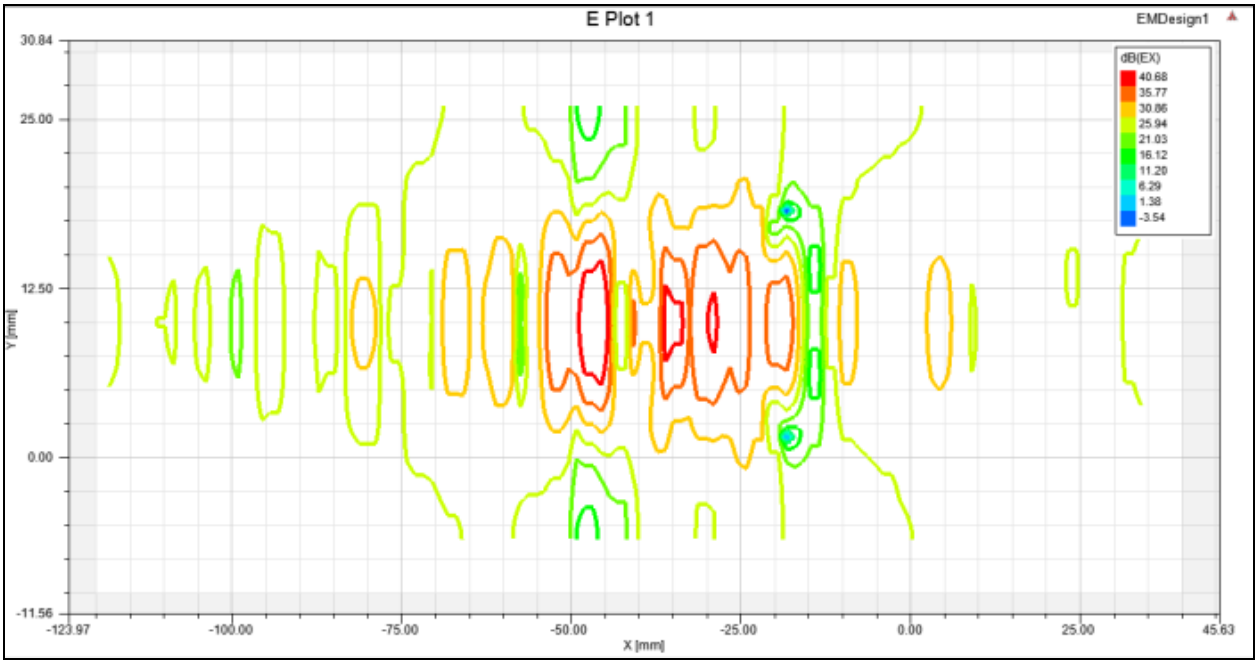
To analyze the radiated fields associated with a design, define a radiation surface over which the fields are calculated. Such a surface can be a boundary radiation surface, or a custom radiation surface that you define as an object face.

To plot near fields on a plane in HFSS 3D Layout:

1. First define a near field setup. From the **Project Tree**, right-click **Radiation** and select **Insert Near Field Setup** > **Plane**
2. From the **Near Field Plane Setup** window that opens, click **OK**. A new Plane setup appears in the **Project Tree**.
3. From the **Project Tree**, right-click **Results** and select **Create Near Fields Report,** then select **Rectangular Contour Plot** or **Data Table**.
4. From the Report window that opens, select an appropriate setup from the **Solution** menu and select a plane setup in the **Geometry** menu. Enter "x" for the **Primary Sweep** and "y" for the **Secondary Sweep**, then click **New Report**. The plot appears and a new radiation

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setup with the name of the selected Plane is created in the **Project Tree**.



For more information, see *Getting Started with HFSS: A Ball Grid Array IC Package: Fields Post Processing*.

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:

To access the **Solution Data** window, in which the information above can be accessed, do one of the following:

- Click **HFSS 3D Layout**> **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.

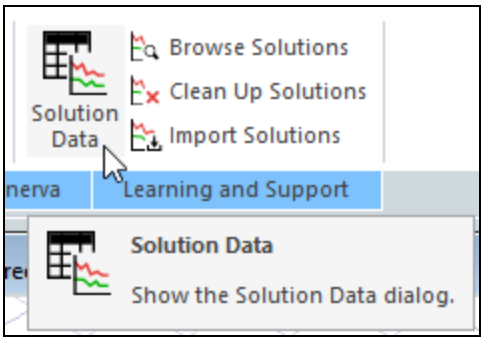
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing a Solution Profile

At any time during or after the solution process, you can examine the computing resources or profile data that were used by Ansys Electronics Desktop solvers during the analysis. The profile data is essentially a log of the tasks performed by Ansys Electronics Desktop during analysis. The log indicates the length of time each task took and how much physical and disk memory was required.

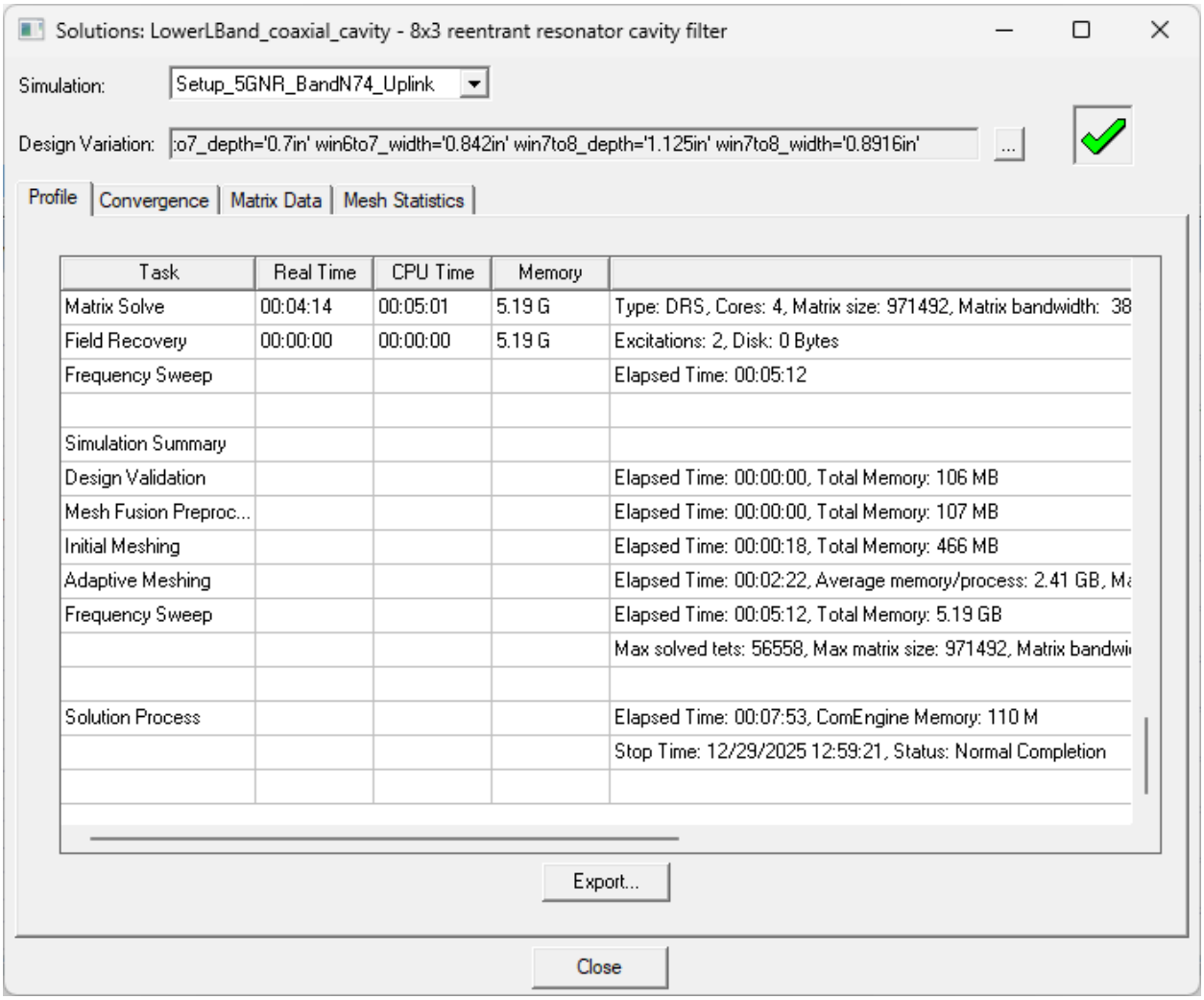
1. In the project tree, right-click a solution setup, and select **Profile**.

   The **Solutions** dialog box appears, with the **Profile** tab selected. You can also click the **Solution Data** icon on the ribbon to open the **Solutions** dialog box, and then select the **Profile** tab.



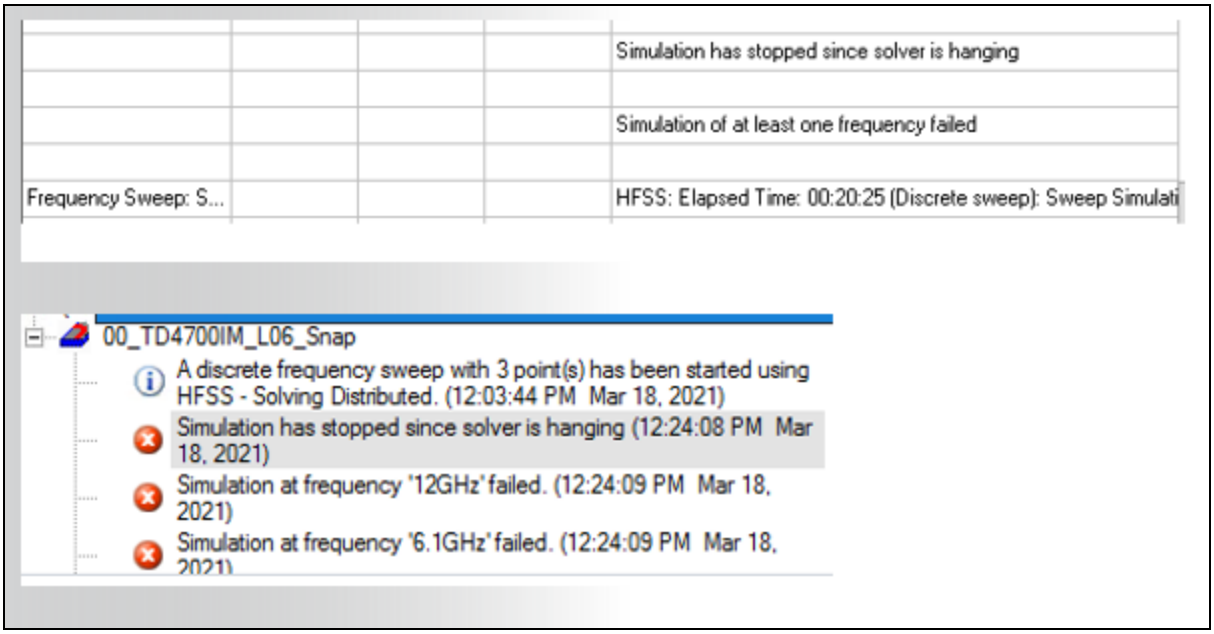
   The displayed data depends on the type of problem and solution setup.

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 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 Start, various Mesh tasks, Simulation Setup, Port Adaptation, Adaptive Pass tasks, including simulation setup, Matrix Assembly, Solver tasks, and Field Recovery, Sweep tasks, and Solution Process summary and Totals for time.
Real Time	The difference in time between the start of the task and the end of the task (elapsed time).
CPU Time	The amount of CPU time required to perform the task.
Memory	The peak amount of physical memory (RAM) used by the individual executable running the task. The memory is freed for other uses after each task is complete.
Information	General information about the solution (for example, the number of tetrahedra used in the mesh, disk use, solver information, sweep information, and totals).

To export the profile data:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Open the **Solutions** dialog box and select the **Profile** tab.
2. Click **Export Profile**.

   This opens a file save dialog box that lets you provide a file name and location.

3. Click **Save**.

   The data is saved in a text file with a .prof extension.

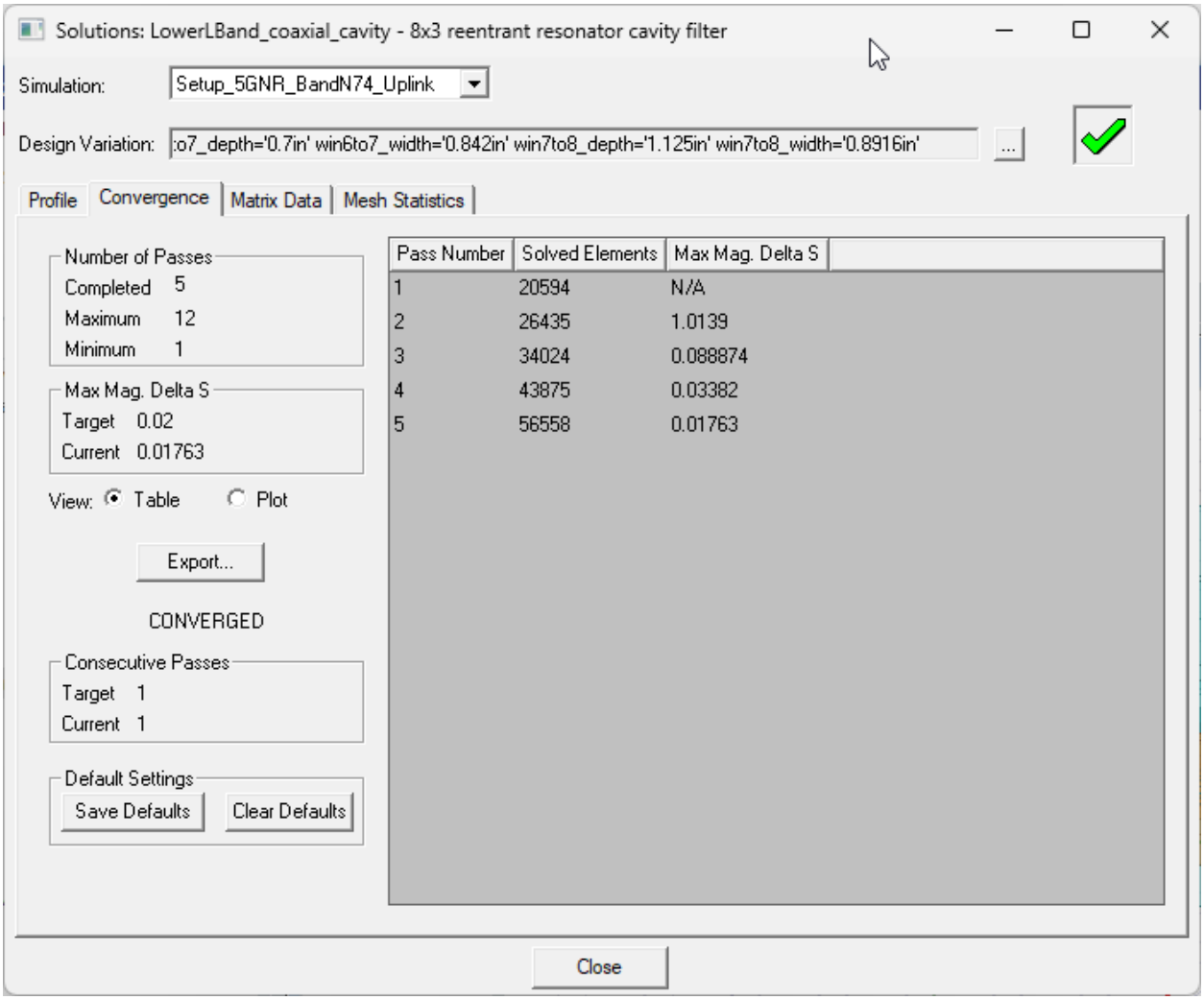
Viewing Convergence Data

To view an adaptive solution's convergence information, either during or after the solution process:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

   The **Solution Data** window appears. The **Convergence** tab is selected. You can also select the **Results** tab of the ribbon, click the **Solution Data** icon, and click the **Convergence** tab when the **Solution Data** 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. If you have run multiple simulations, from the **Simulation** list, select the solution setup for which you want to view convergence data.

   By default, the most recently solved solution is selected.

3. Under the **Convergence** tab, depending on your design setup, you can review the following convergence data:
   - Whether the solution is converged or not converged. The dialog displays 'CONVERGED' if last adaptive pass meets the convergence criterium.
   - Number of adaptive passes completed and remaining.
   - The Solved Elements at each adaptive pass, which includes solve inside tetrahedra, and, for projects using IE Regions for metal objects, also includes the number of solved IE surface triangles.
   - Maximum magnitude of delta S between two passes.

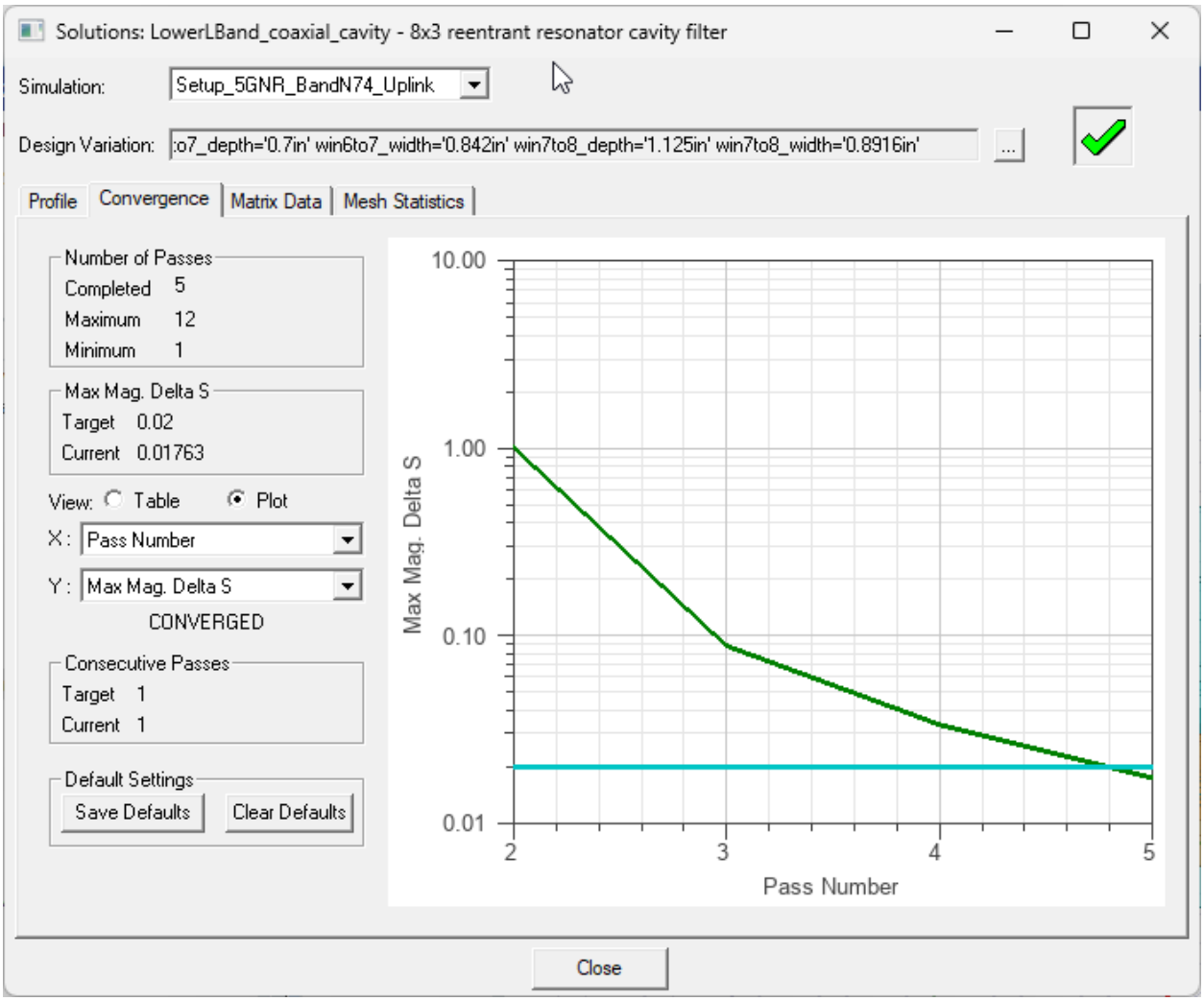
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Maximum delta Energy between two passes.
- Magnitude margin between passes.
- Phase margin (deg) between passes.
- Maximum delta frequency between passes.

  If for the Solution Setup, you elected to **Use Matrix Convergence**, and selected specific table entries for the Magnitude and Phase, the **Convergence** tab also shows the following values with the Magnitude Margin and Phase Margin:

- Max Delta (Mag S)
- Max Delta (Phase S)

4. Select **Table** to display the convergence data in table format or **Plot** to plot the convergence data on a rectangular (X - Y) plot.



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 select **Save Defaults**, the settings for Table vs. Plot, and for Plot, the X and Y axis selections, those settings apply, based on the design-type, each time you open the desktop. You can also select **Clear Defaults** to reset to the original settings.

> **Note:**
>
> If you receive a message that the eigenmodes have not converged, it may indicate that the existing mesh is too coarse. You may need to refine the mesh.

Viewing the Number of Completed Passes

At any time during the solution process, you can view the number of adaptive passes (solve — error analysis — refine cycles) that have been completed and that have yet to be completed. When the solution is complete, you can view the number of adaptive passes that were performed. If the solution converged within the specified stopping criteria, fewer passes than requested may have been performed.

To view the number of passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The number of completed and remaining passes is listed in the **Number of Passes** area.

Viewing the Max Magnitude of Delta S Between Passes

For solutions with ports.

At any time during or after the solution process, you can view the maximum change in the magnitude of the S-parameters between two consecutive passes. This information is available after two or more passes are completed.

To view the maximum magnitude of delta S between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Max. Mag. Delta S** column lists the maximum magnitude of delta S from one pass to the next.

The **Max. Mag. Delta S** area lists the target change in magnitude of delta S and the change in magnitude of delta S between the last two solved passes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Delta S is computed on the appropriate S-parameters - modal or terminal - after the S-parameters have been deembedded and renormalized.

Note:

You can renormalize mathematically, without having to re-solve, by accessing the postprocessing tab on the port definition panel and deselecting the **Deembed** selection box.

Viewing the Delta Magnitude Energy

For designs with voltage sources, current sources, or incident waves. Not applicable to designs with ports.

At any time during or after the solution process, you can view the difference in the relative energy error from one adaptive pass to the next. The change in the magnitude of delta energy is available after two or more passes are completed.

To view the delta magnitude E between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Delta Mag. Energy** column lists the delta energy from one pass to the next.

The **Delta Mag. Energy** area lists the target change in delta energy and the change in delta Energy between the last two solved passes.

Plotting Convergence Data

To display convergence data vs. pass on a rectangular (x - y) plot:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

2. In the lower-left corner of the window, select **Plot** as the view type.
3. Select the data you want to plot on the x-axis from the **X** drop-down menu.
4. Select the data type you want to plot on the y-axis from the **Y** 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.

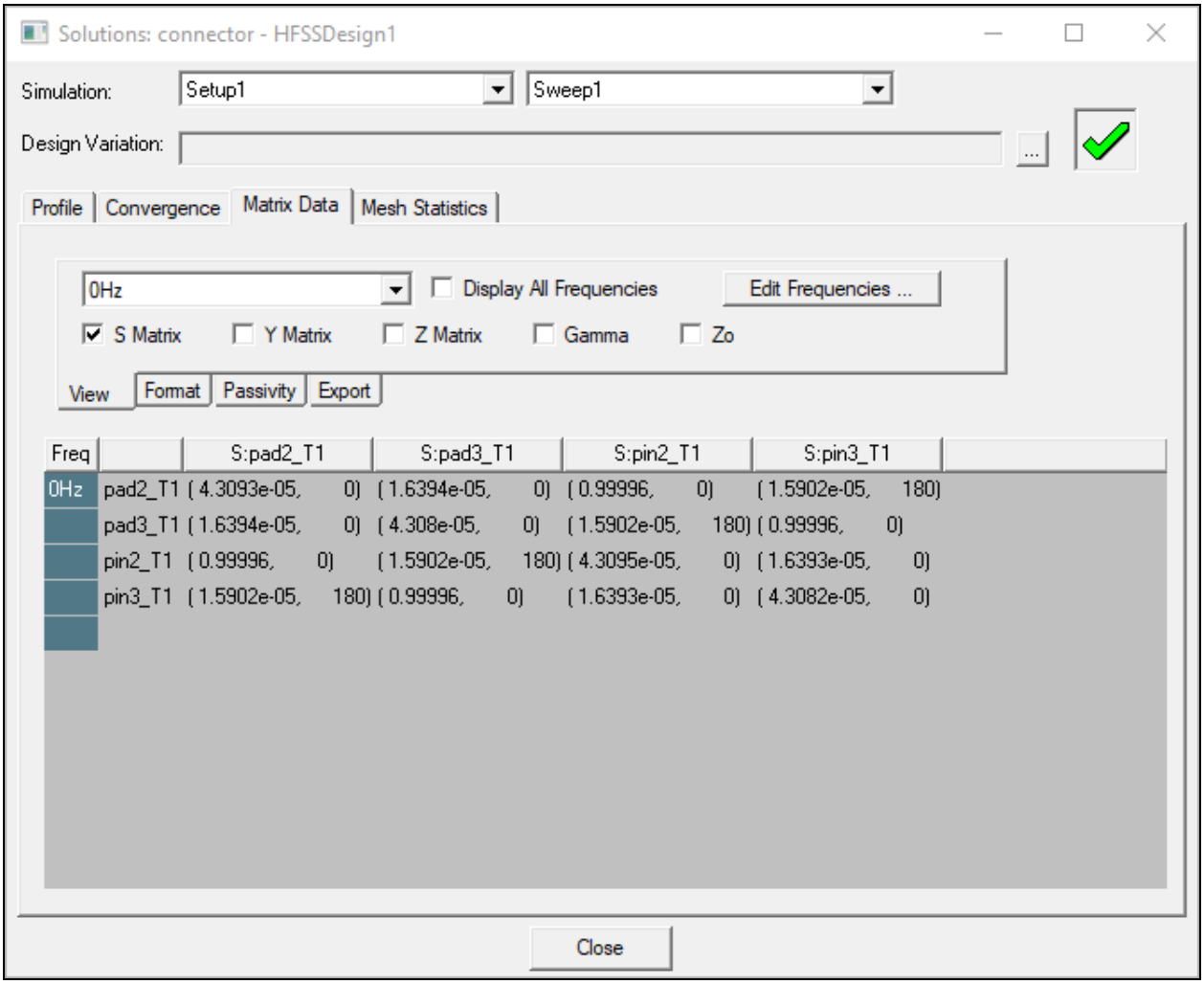
The x -y plot appears in the view window.

Viewing Matrix Data

To view matrices computed for the S-parameters, impedance, and propagation constants during each adaptive, non-adaptive, or sweep solution:

1. In the **Project** tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu, or on the **Results** tab of the ribbon, click **Solution Data**.

   The **Solution Data** dialog box appears. The **Matrix Data** tab is selected.



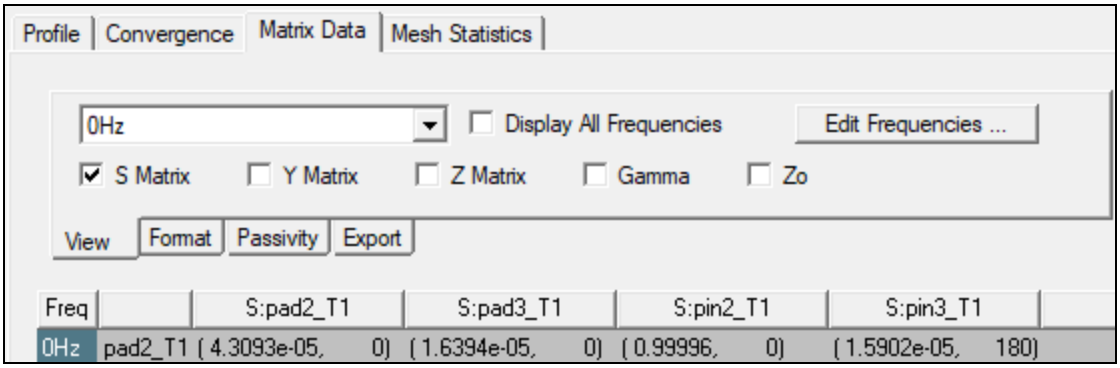
2. Select which pass to view in the drop down list to the right of the **Simulation** list. To select a specific adaptive pass:
   a. Choose **Adaptive Pass**.
   b. On the **View** tab, select the pass you want from the **Pass** list.

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 design variations exist, in the **Design Variation** text box, specify the design with the matrices you want to view.

   Optionally, choose a design variation solved during an Optimetrics analysis from the **Set Design Variation** dialog box. This lists all the solved variations in the design. This dialog box is accessible from the **Solution Data** window by clicking the ellipsis button on the right of the Design Variation field, and via the **[solver]** > **Results** > **Apply Solved Variation** command.

4. In the **Simulation** drop-down menu, click the solution setup and solved pass - adaptive, single frequency solution, or frequency sweep - for which you want to view matrices.
5. Select the type of matrix you want to view: **S-matrix**, **Y-matrix**, **Z-matrix**, **Gamma**, or **Zo** (characteristic impedance.) The available types depend on the solution type.



6. Select the solved frequencies to display:
   - To display the matrix entries for all solved frequencies, select **Display All Freqs**.

     If **Display All Freqs** is enabled, the drop-down frequency select list is disabled.

   - To show the matrix entries for a selected solved frequency, ensure that **Display All Freqs** unchecked and use the drop-down menu to select the solved frequency for which you want to view matrix entries. You can use a scroll bar for selecting from long frequency lists.

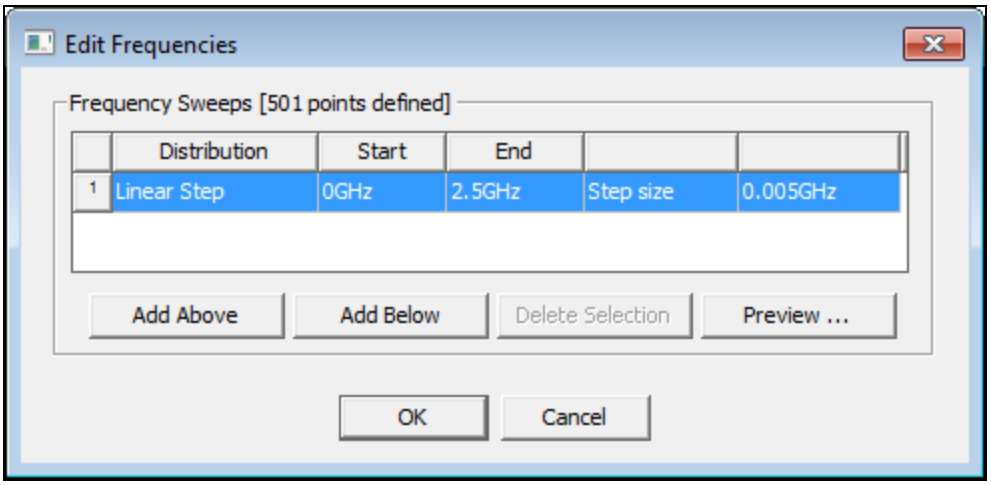
     For adaptive passes, only the solution frequency specified in the **Solution Setup** dialog box is available. For frequency sweeps, the entire frequency range is available.

   - To insert or delete one or more displayed frequencies, click **Edit Frequencies**.

> **Note:**
>
> This command is only available if the sweep type is Fast or Interpolating.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

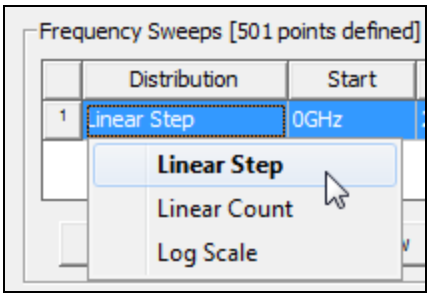
Clicking **Edit Freqs** displays the **Edit Frequencies** dialog box. The current values are displayed in a table.



Note:

Changes to the Start value and End value cannot be outside of the initial range. No message is issued: rather the range is implicitly restricted.

Use the **Add Above** or **Add Below** button to add a new frequency to the table above or below the currently selected value. If no value, or the start value is selected, the new frequency repeats the current Start value and increments the count at the start a new row. You can select the type of distribution for a new Row by selecting from the drop down.

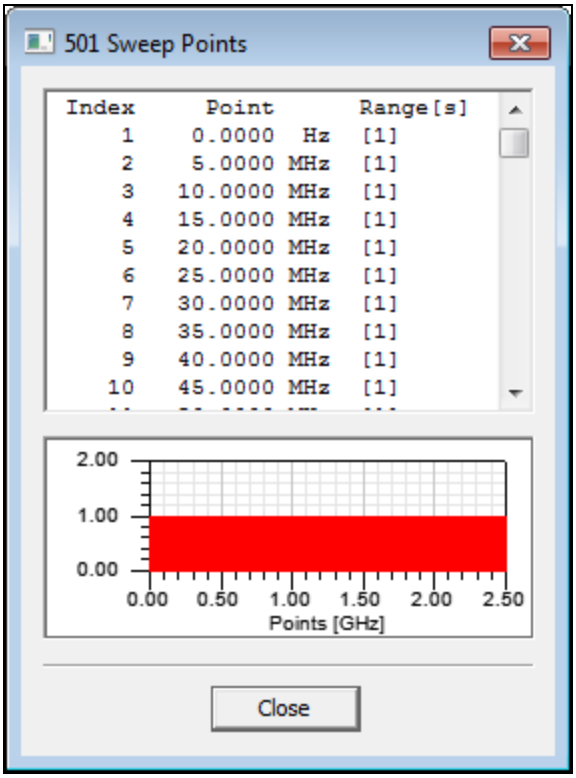


You can then edit the **Start** and **Stop** fields as appropriate for the new row.

The **Delete Selection** button enabled only if a value is selected. **Delete Selection** removes the selected value and decrements the Number of Values field.

The **Preview** button opens a dialog showing the defined 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.



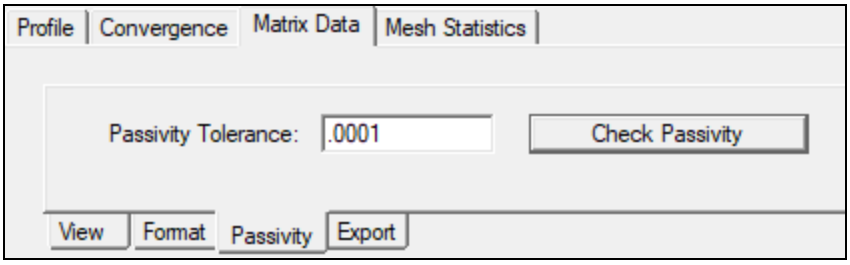
7. Select the display format — On the **Format** tab, select from the drop down menu which has the options **Magnitude/ Phase (deg)**, **Real/ Imaginary**, **dB/Phase (deg)**, **Magnitude**, **Phase (deg)**, **Real**, **Imaginary**, and **dB** — in which to display the matrix information and then in the next drop down menu, select **Terminal Data** or **Differential Pairs**.
8. Click **OK** to apply the changes to the **Solutions** dialog **Matrix Data** tab and close the **Edit Sweep** dialog box, or **Cancel** to close the dialog without applying the changes.

   If you choose to export the matrix data for the Fast or Interpolating sweep after modifying the frequencies in the **Edit Frequencies** dialog box, only those frequencies displayed under the **Matrix Data** tab will be exported.

   The data is displayed in the table. By default, wave ports are listed in alphabetical, then numerical order, just as they appear in the excitation tree. To change the port order, change setting for Default Matrix sort order in the HFSS General options. You may also want see how you can Reorder Matrix Data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Optionally, select the **Passivity** tab and **Check Passivity**.



This passivity check tests whether the S-parameter data from HFSS is passive or not. If the S-Matrix is not passive at one or more frequencies, this check displays a dialog saying that the solution is passive for all frequencies or that identifies the worst frequency violation and identifies the passivity in that case. A uniform renormalization of 50 ohms is performed on the solution data for Passivity checking.

Selecting the Matrix Display Format

The **Solutions** dialog offers a range of formats for the Matrix display. The available formats depend on the matrix type being displayed. When selected, dB formatting only applies to S - matrix data, even if other matrix types are displayed. The column heads in the display identify the format for the matrix type. You can display matrix data in the following formats.

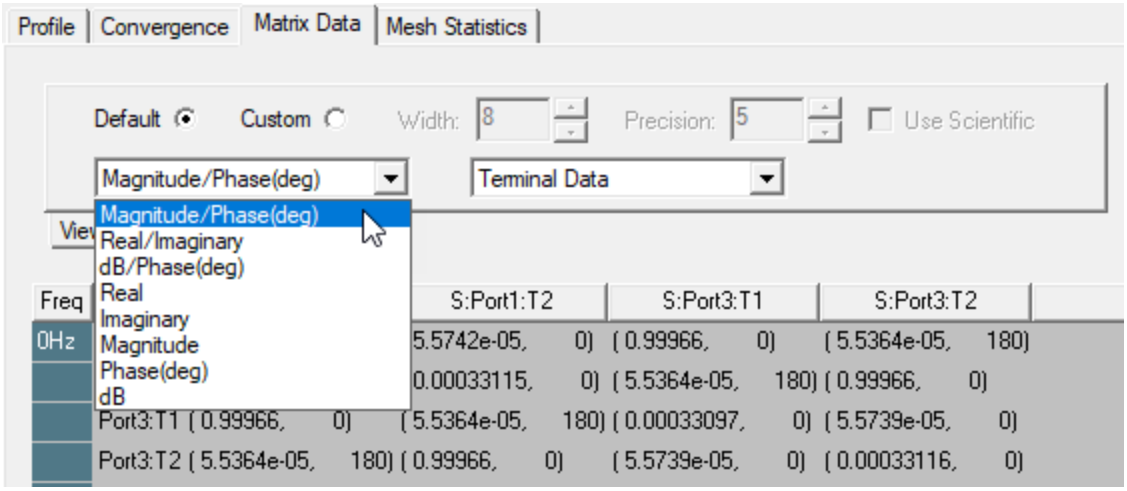
Magnitude, Phase (deg)	Displays the magnitude and phase (in degrees) of the matrix type.
Real, Imaginary	Displays the real and imaginary parts of the matrix type.
dB, Phase (deg)	Displays the magnitude in decibels and phase in degrees of the matrix type.
Magnitude	Displays the magnitude of the matrix type.
Phase (deg)	Displays the phase in degrees of the matrix type.
Real	Displays the real parts of the matrix type.
Imaginary	Displays the imaginary parts of the matrix type.
dB	Displays the magnitude in decibels of the matrix type.

After you select a format, choose whether to display terminal data or differential pairs.

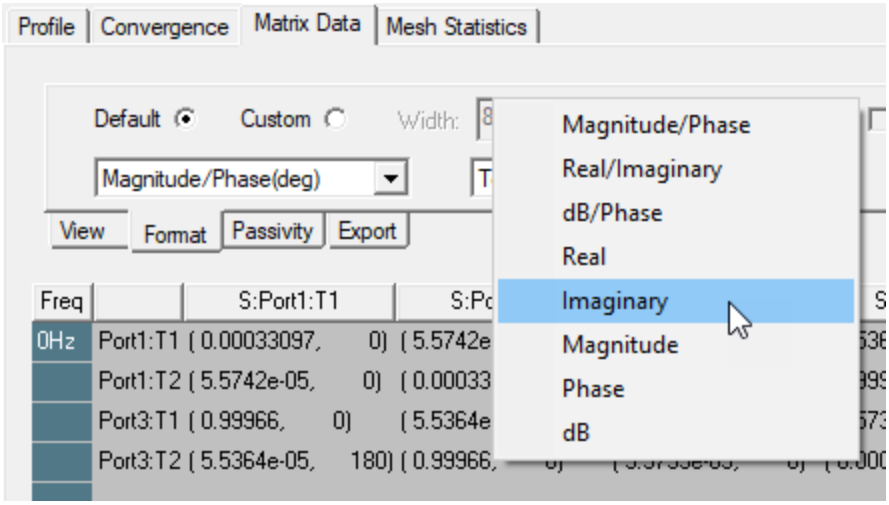
In the **Matrix Data** tab of the **Solutions** window, open the **Format** tab to select the display format in one of two ways:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the drop-down menus under the formats:



- right-click the column headings to display a pop-up format menu.



- Selecting from this menu also adds the name of the selected format to the column head, if the format is anything other than dB.

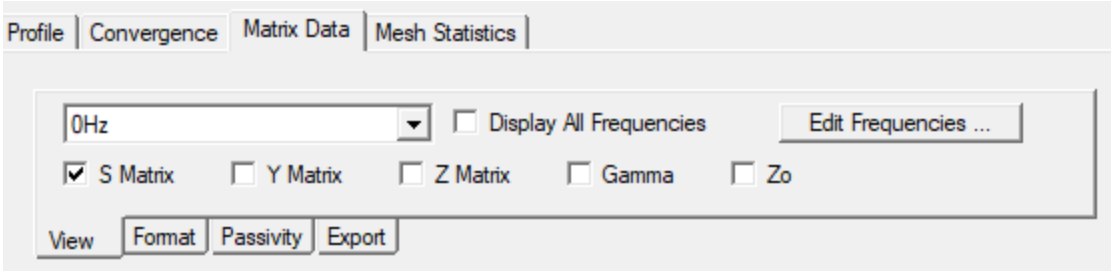
Freq		S:pad2_T1 (Imag)	S:pad3_T1 (Imag)	S:pin2_T1 (Imag)	S:pin3_T1 (Imag)
5 (GHz)	pad2_T1	-0.10414	0.056068	-0.75051	-0.1483
	pad3_T1	0.056068	-0.104	-0.14825	-0.75036
	pin2_T1	-0.75051	-0.14825	-0.16368	0.064232
	pin3_T1	-0.1483	-0.75036	0.064232	-0.1635

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

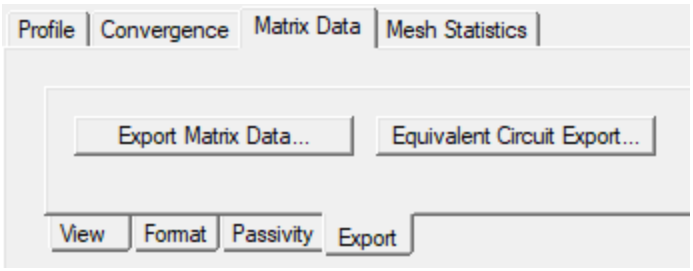
Exporting Matrix Data

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

The **Solution Data** window appears. The **Matrix Data** tab is selected. You can also access the **Solution Data** window from the **Results** tab on the ribbon by clicking the **Solution Data** icon.

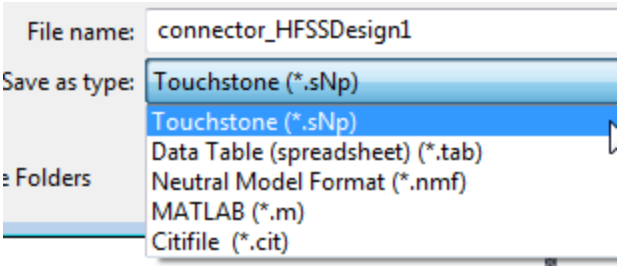


2. Select the type of matrix you want to view: **S-matrix**, **Y-matrix**, **Z-matrix**, **Gamma**, or **Zo** (characteristic impedance.)
3. Select the **Export** tab and click **Export Matrix Data**.



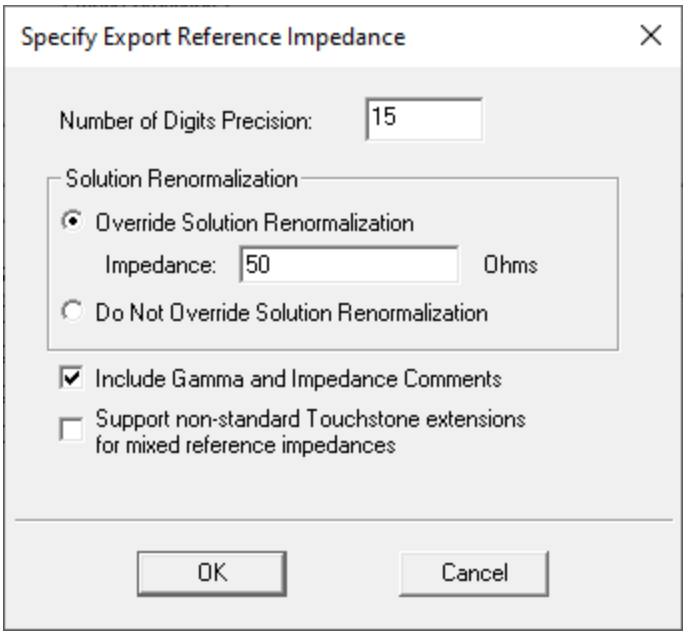
A file browser appears.

4. Type the name of the file you are exporting to in the **File name** text box.
5. Select one of the following file formats from the **Save as type** 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.

Format	Type	Description
(spreadsheet) *.tab	Data Table	A text file in which the elements of the S-matrix are arranged in a series of columns that are tab-separated and include a first row of headings. The file may be imported into a spreadsheet or similar utility.
***.sNp**	Touchstone	A Touchstone S-parameter file in which the number of ports is indicated by *n*. For example, a Touchstone file with one port would have the file extension .s1p. When you export this format, you are presented with a dialog.



You can specify:

- **Number of Digits Precision** (Default 15)
- **Override Solution Renormalization**. This refers to overriding the renormalization impedances that may have been asked for in the port setup. If so, the export renormalizes impedance
- **Do Not Override Solution Renormalization.** This leaves any port setup renormalization options in place, and grays out the Impedance setting on the dialog.
- **Include Gamma and Impedance Comments**
- **Support for non-standard Touchstone extensions for mixed reference impedances**. Writing gamma and Z_0 is not Touchstone standard, but rather an HFSS "feature" that only HFSS will

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		recognize.
		If you want to export raw S-Parameter data for later use, you may choose to not renormalize the solution.
		If all ports and associated modes/terminals are normalized to the same impedance and you choose **Do Not Override Solution Renormalization** during export, the Touchstone file header will indicate the normalized impedance.
		The comment header in the Touchstone file lists the port and mode numbers to show which column contains which port name (in case of confusion between alphabetical and force repriority ordering of ports and associated modes).
		`! Touchstone file exported from HFSS 2026.1`
		`! File: D:/Program Files/AnsysEM/v242/Examples/HFSS/Signal Integrity/package.aedt`
		`! Generated: 1:56:02 PM Apr 01, 2020`
		`! Design: HFSSDesign1`
		`! Project: package`
		`! Setup: Setup1`
		`! Solution: Sweep1`
		`!`
		`# GHZ S MA R 50.000000`
		`! Terminal data exported`
		`! Port[1] = P01__NET179__T1`
		`! Port[2] = P02__NET178__T1`
		`! Port[3] = P03__NET178__T1`
		`! Port[4] = P04__NET179__T1`

***.nmf**	Neutral Model format	Neutral Model file format defined by the MAFET Consortium. When you save to this format, if the design includes variables, you are presented with the NMF Parameters dialog that lets you select which variables to select as parameters in the NMF file. Non-selected variables will be give the constant value shown. You are then presented a dialog that lets you specify the number of digits precision for the renormalizing impedance.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***.m**	MATLAB	The Mathworks' MATLAB file in which the elements of the S-, Y-, or Z-matrix are arranged in a series of rows. MATLAB only allows creation of complex numbers by specifying real and imaginary parts, not by specifying magnitude and phase.
***.cit**	Citifile	Common Instrumentation Transfer and Interchange file format. It is an ASCII format defined by instrument and CAE designers.

> **Note:**
>
> For Touchstone files, you no longer see a **Combine Sweeps** option on the **Export Network Data** solution dialog box. This is because the current Sweep setup allows you to define multiple sweeps for a single simulation, meaning any needed combining is defined with the sweep setup.

6. Click **Save**.

The data is exported to the file.

- By default, wave ports are listed in alphabetical, then numerical order, just as they appear in the excitation tree. You can change this order to creation order and back without invalidating the solution on the HFSS Options dialog box.
- If you select Touchstone format, you are first presented with a dialog in which you can specify the Number of Digits precision, and Override the Solution Renormalization. If so, you can specify the export renormalizing impedance (an integer value). Here you also can specify and whether to include **Gamma and Impedance Comments**, and **Number of Digits Precision** (Default 15).
- If you select Neutral File Format, you are presented with a **Specify Export Renormalizing Impedance** dialog that lets you specify the Number of digits precision for the save file.

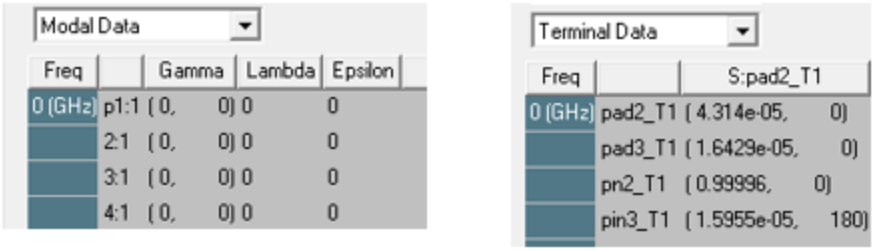
> **Note:**
>
> If you modify the display of solved frequencies in an Interpolating or Fast sweep under the **Matrix Data** tab (by clicking **Edit Freqs** and then modifying the values in the **Edit Sweep** dialog box,) only those frequencies listed will be exported to the file.

Renaming Matrix Data

In the project tree, you can right-click a port excitation to rename it.

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 rename a port excitation, the associated Matrix data is reordered so that it can be presented in the same manner. The reordering is done to match the tree-sort order presented for the ports (renamed matrix data is reordered so that alphabetic values appear before numeric values).



Exports of the matrix data are ordered in the same manner. This reordering is conducted as part of post processing and does not force a re-solve.

Reordering Matrix Data

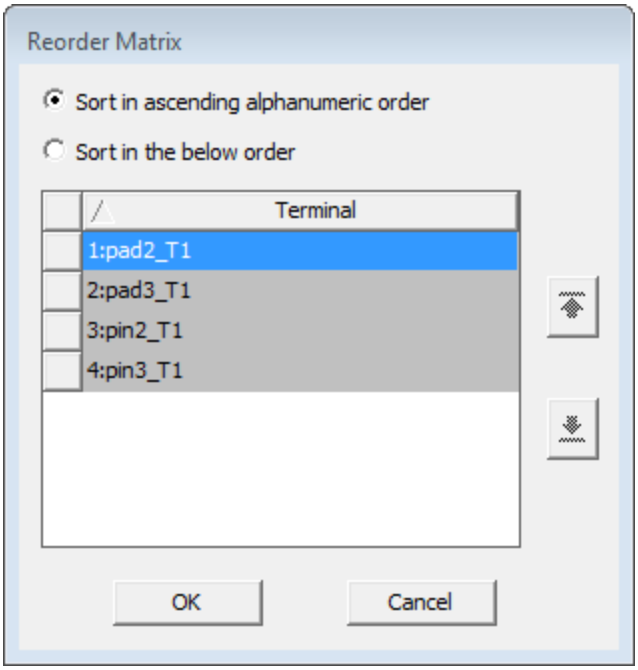
HFSS lets you reorder the matrix data as a post-processing step.

1. To re-order the matrix data, either as the default ascending alphanumeric order, or a user specified order, click **HFSS>Excitations>Reorder Matrix**, or right-click Excitations in the Project tree, and click **Reorder Matrix** on the shortcut menu.

This displays the **Reorder Matrix** dialog. It lists the ports for a modal solution, or the terminals for a terminal solution. For Transient solutions, the dialog displays separate lists for

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

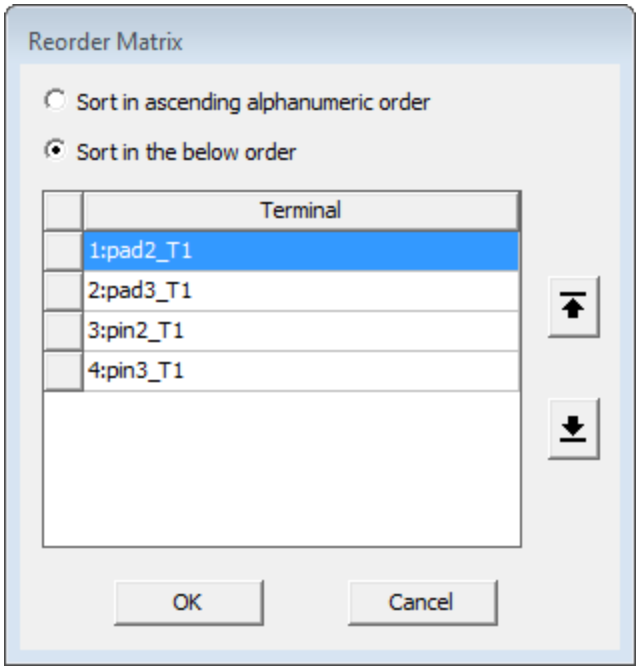
Active and Passive ports.



2. You can select the radio button for **Sort in ascending alphanumeric order**, or **Sort in the below order**.

If you select **Sort in the below order**, you can select any port or terminal to enable the up and down arrow keys. Use the keys to move the ports into any desired order. In the case of Transient, the arrow keys operate only within a partition, that is only within Active or Passive lists.

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 Ctrl+click to select multiple arbitrary ports, or hold Shift to select a range of ports. Clicking outside the list deselects all selections.

Note:

If there are differential pairs, the sort order is still specified) in terms of the underlying terminal names, but the entries that make up the pair should appear in the appropriate sort location for the terminals that are used to define them.

3. Click **OK** to accept the order specified.

For Transient solutions, removing an active source will not affect the solve but might affect the matrix order. Adding an active source will require a resolve if that source has not already been solved.

Exporting Equivalent Circuit Data

You can export S-parameter data from a Driven Terminal solution to PSpice, HSPICE, Spectre or Maxwell Spice format. Importing the new data file to PSpice, HSPICE, Spectre or Maxwell Spice will enable you to include wave effects in the circuit simulations. You can also export a W-Element model for a port.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

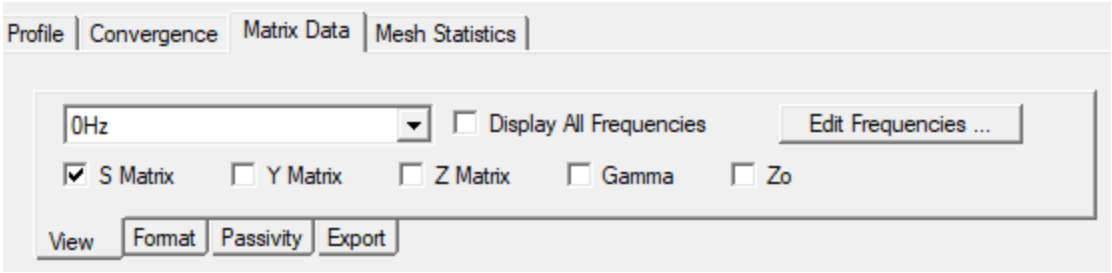
Note:

You must have a frequency sweep solution and five or more frequency points to successfully export an equivalent circuit data file. See the *Choosing Frequencies for Full-Wave SPICE* topic of the help for suggestions about the frequency range of the sweep.

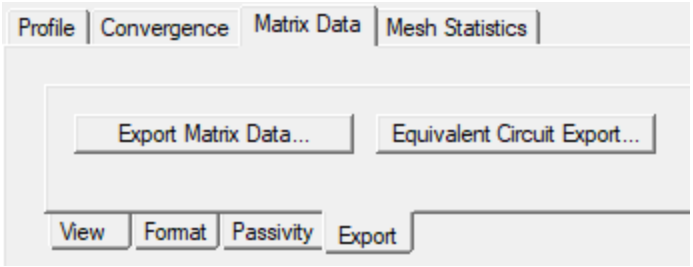
Certain discrete sweeps permit Full-Wave SPICE exports. It is allowed if the discrete data is evenly spaced, includes DC, and has at least 500 frequency points.

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

The **Solution Data** window appears. The **Matrix Data** tab is selected. You can also access the **Solution Data** dialog from the **Results** tab on the ribbon by clicking the **Solution Data** icon.



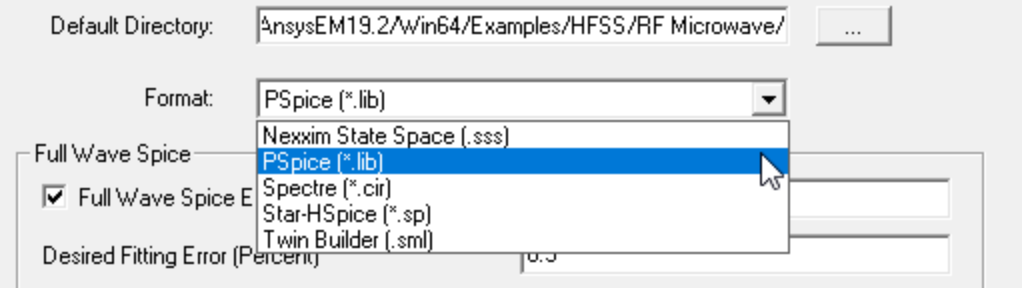
2. Select the **Export** tab and Click **Equivalent Circuit Export**.



The **Equivalent Circuit Export 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.

3. Type the name or browse to the directory in which you want to store the data.



4. Click one of the following formats in the **Format** list:

   PSpice (*.lib)
   Nexxim State Space (.sss)
   Spectre (*.cir)
   Star HSpice (*.sp)
   Twin Builder (.sml)

Your format selection affects the options available under **Full Wave Spice Export**. When Twin Builder format is requested, both *.sml and *.png are created. The latter contains the image in GIF format.

> **Note:**
>
> The Export to Twin Builder here does not use the same settings as Nexxim or Network Data Explorer. If you intend to export to Twin Builder, you should use the NdExplorer.

5. If the **Full-Wave Spice Export** check box is enabled, you can select it. Checking the box enables the text field for the file name, and depending on the format selection, other options may be enabled.
6. Desired Fitting Error (percent) has a default value of 0.5. You can edit this.
7. Maximum Order has a default value of 10000.
8. HFSS supports Full Wave Spice Export from a driven modal design as long as all ports have exactly one mode each. However, HFSS does not support definition of differential pairs in a driven modal design.
9. By default **Use Common Ground** is checked and produces circuit models with a "common" (suppressed) ground terminal. Uncheck Use Command Ground to apply one negative reference terminal per port.
10. Optionally, select **Enforce Passivity**. Selecting this enforces passivity in the output file. Passive devices can only dissipate or temporarily store energy, but never generate it.

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 check passivity from the **Matrix Data** tab using the Check Passivity button.)

This option is useful in cases where the transient simulation fails due to passivity violations in the circuit model. This circuit model is based on fitting a rational function to the S-parameter data computed by the field solver. Small errors in the data fitting can result in non-passive behavior. Selecting the **Enforce Passivity** option will take more CPU time, but ensures that the resulting model will be passive. The **Enforce Passivity** check box uses "Iterated fitting of passivity violations (IFPV)" method to do the passivity enforcement.

The passivity check tests whether the S-parameter data from HFSS is passive or not. For more information see Passivity.

11. Optionally, select **Lumped Element Export (Low Bandwidth)** if you want to save the data as a low-frequency circuit model using simple lumped elements (resistors, capacitors, inductors, and dependent current sources). The low-bandwidth model is only going to be accurate in a limited frequency range around the adaptive solution frequency

    This option is not enabled for Spectre export.

12. Optionally, select **Partial Fraction Expansion for Matlab** if you want to specify a file that expands the partial fractions for use in MATLAB. The partial fractions involved describe the frequency response of the low-bandwidth model from the previous step.
13. You can also select **Combine Sweeps** to select and combine available sweeps into a single output file.

    By option, for Nexxim State Space, Spectre, and Star-HSpice, you can Export W-Element Data.

14. Click **OK**.

    The S-matrices are written to the data file that you specified in the equivalent circuit data format.

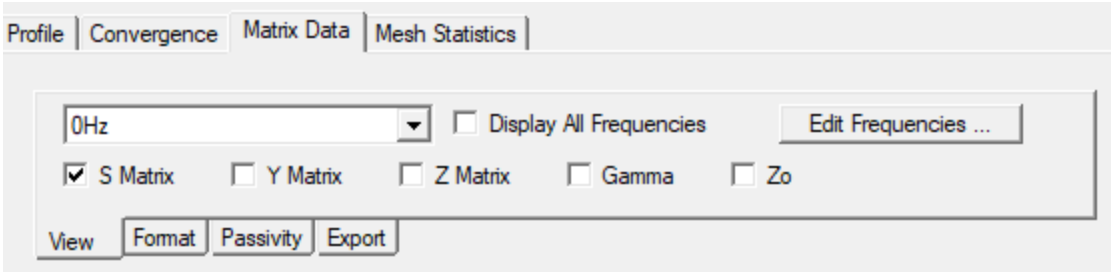
Exporting W-element Data

It is possible to extract a W-element model for a port. This W-element model can be used in a SPICE model to represent a length of transmission line of the same cross-section as the port. A W-element model can be extracted for a port only solution and for a full 3D solution.

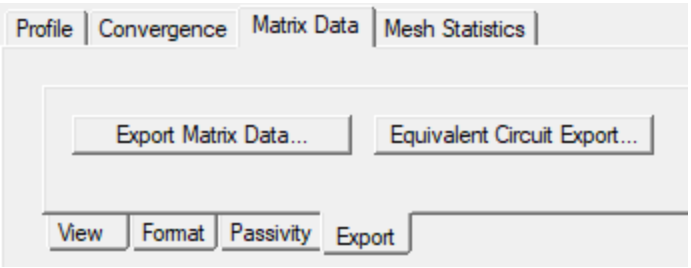
1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

   The **Solution Data** window appears. The **Matrix Data** tab is selected. You can also access the **Solution Data** dialog from the **Results** tab on the ribbon by clicking the **Solution Data** icon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

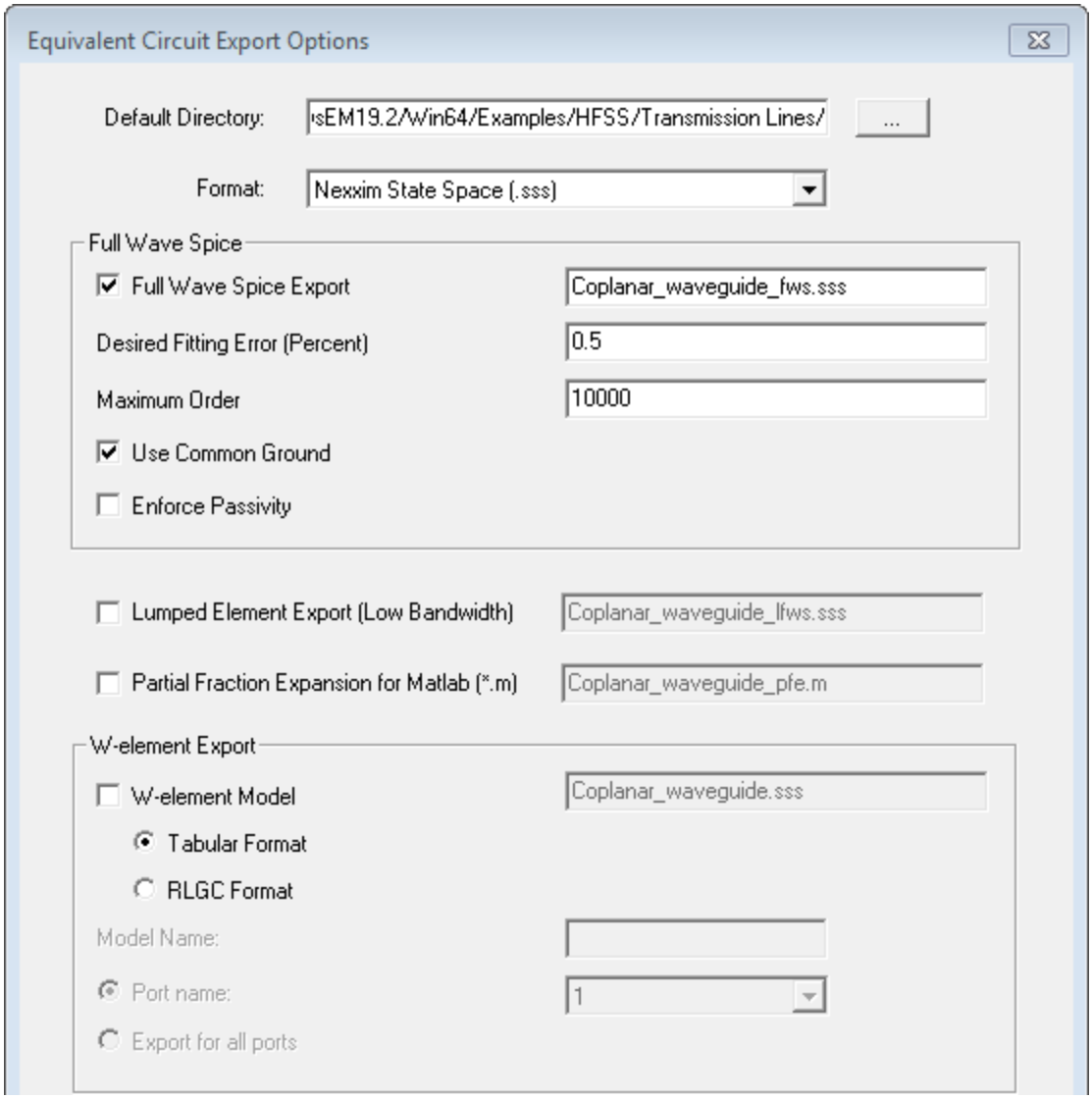


2. Select the **Export** tab and click **Equivalent Circuit Export**.

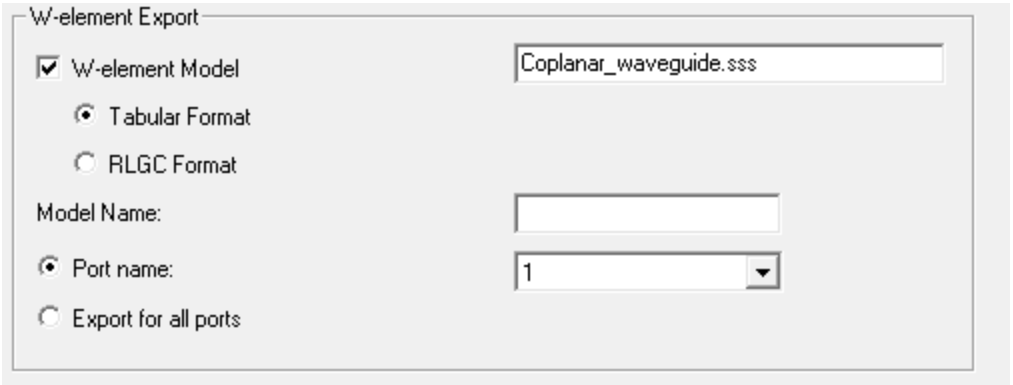


The **Equivalent Circuit Export Options** dialog box appears. If you select the format as Nexxim State Space, Spectre, or Star-HSpice, at the bottom of the dialog you see the W-element model check box enabled.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the W-element model check box to enable the W-element fields.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. The W-element model name field has the project name by default. You can change this if desired.
5. Choose the format as Tabular Format (the default) or RGLC format for W-element export.

Tabular Format: provides a unique RLGC model for each frequency in the solution.

RLGC Format: provides a RLGC fit over a frequency range based on Ro, Lo, Go, Co, Rs and Gd parameters.

> **Note:**
>
> For the RLGC Format, if only a single frequency solution is selected (e.g., LastAdaptive) then Rs and Gd parameters are ignored.

6. In the **Model name** field, provide a model name.
7. Either select the port from the **Port Name** pull down, or, to export a W-element model for all ports, select the **Export for All Ports** radio button.
8. Click **OK**.

The W-element model is written to the data file that you specified.

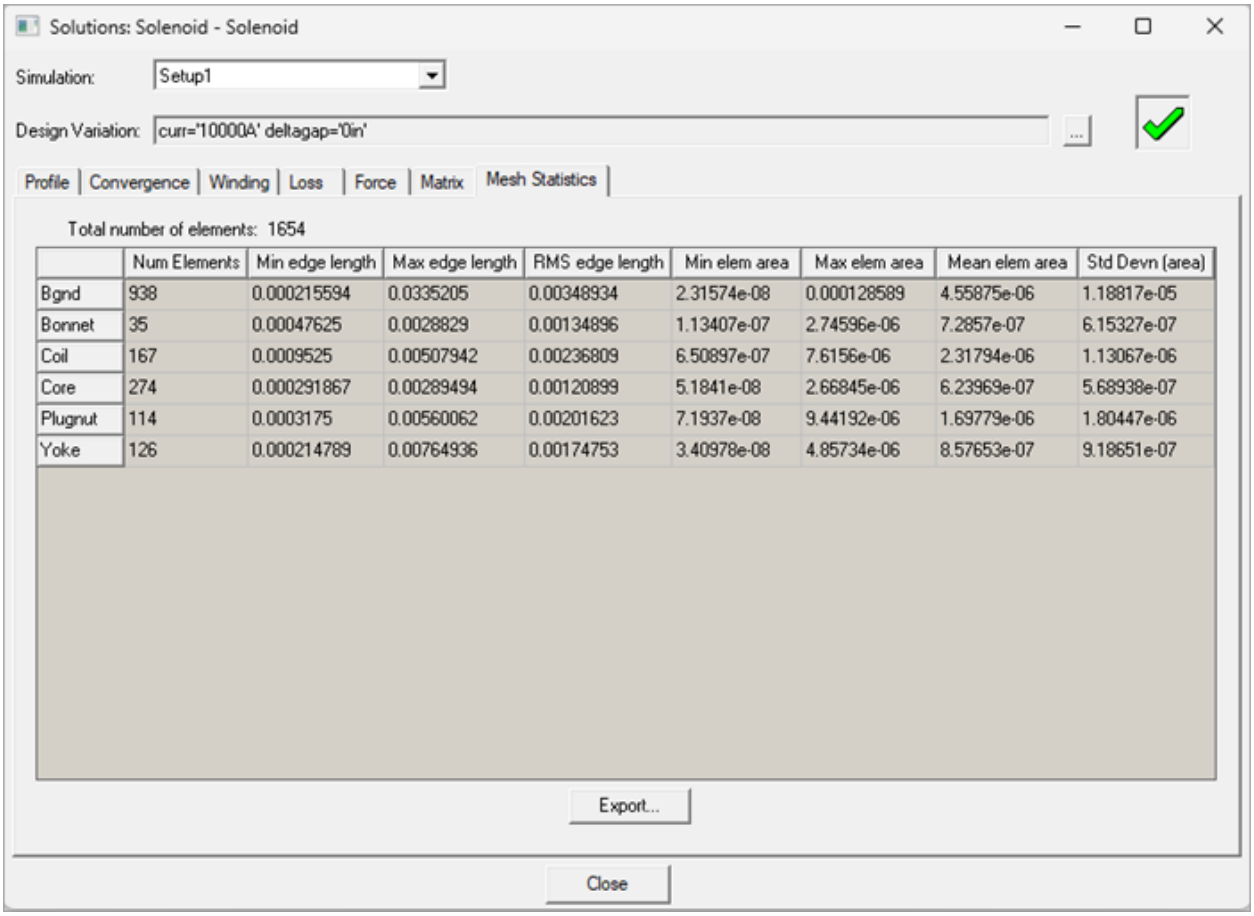
Viewing Mesh Statistics

To view an adaptive solution's mesh information, either during or after the solution process:

1. In the Project Tree, right-click the solution setup of interest, and then click **Mesh Statistics**. Alternately, click the **Solution Data** icon on **Results** tab of the ribbon.

   The **Solutions** window appears with the **Mesh Statistics** tab displayed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	Num Elements	Min edge length	Max edge length	RMS edge length	Min elem area	Max elem area	Mean elem area	Std Devn (area)
Bgnd	938	0.000215594	0.0335205	0.00348934	2.31574e-08	0.000128589	4.55875e-06	1.18817e-05
Bonnet	35	0.00047625	0.0028829	0.00134896	1.13407e-07	2.74596e-06	7.2857e-07	6.15327e-07
Coil	167	0.0009525	0.00507942	0.00236809	6.50897e-07	7.6156e-06	2.31794e-06	1.13067e-06
Core	274	0.000291867	0.00289494	0.00120899	5.1841e-08	2.66845e-06	6.23969e-07	5.68938e-07
Plugnut	114	0.0003175	0.00560062	0.00201623	7.1937e-08	9.44192e-06	1.69779e-06	1.80447e-06
Yoke	126	0.000214789	0.00764936	0.00174753	3.40978e-08	4.85734e-06	8.57653e-07	9.18651e-07

The table lists design elements and for each includes: Num Tets, Min edge length, Max edge length, RMS edge length, min tet vol., max tet vol., mean tet vol. and standard deviation (min elem area, max elem area, mean elem area and standard deviation for 2D designs).

If mesh repairs have been performed, two additional columns appear in the table: **Recovered %** and **Repaired %**. These columns indicate the fraction of an object that was successfully recovered, and the fraction that needed some repair.

To sort mesh statistics by column from low to high values or vice-versa, click a column header. This displays a shadowed triangle pointing down to indicate a list ordered from highest to lowest; and a triangle pointing up to indicate a list ordered from lowest to highest. Clicking again inverts the order.

Click on a blank cell above the object list to invert the order of objects. In this case, the cell does not display a directional triangle.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

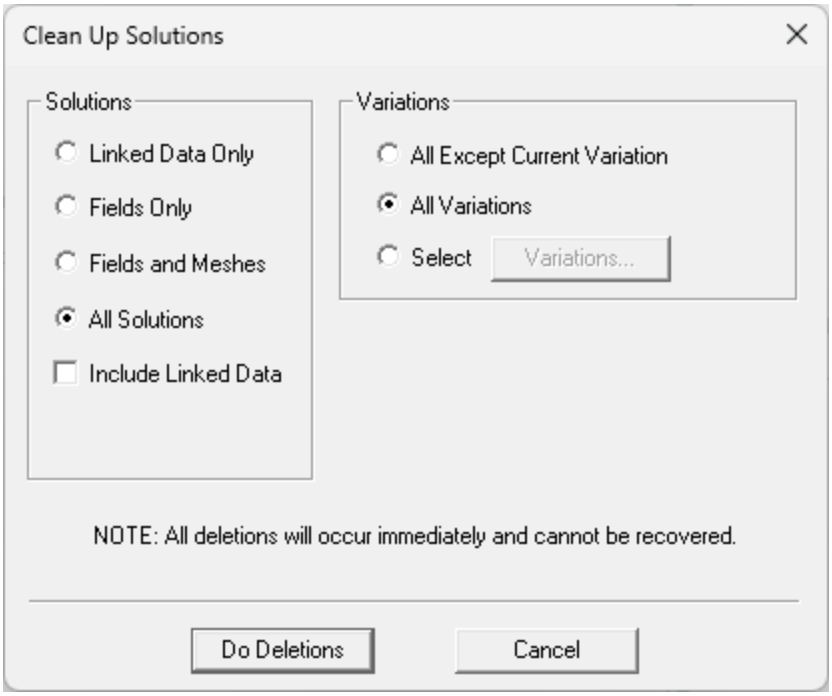
Deleting Solution Data

You can use **Clean Up Solutions** to selectively make deletions, or remove all solutions from the results.

To use **Clean Up Solutions**:

1. Click **HFSS 3D Layout** > **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.

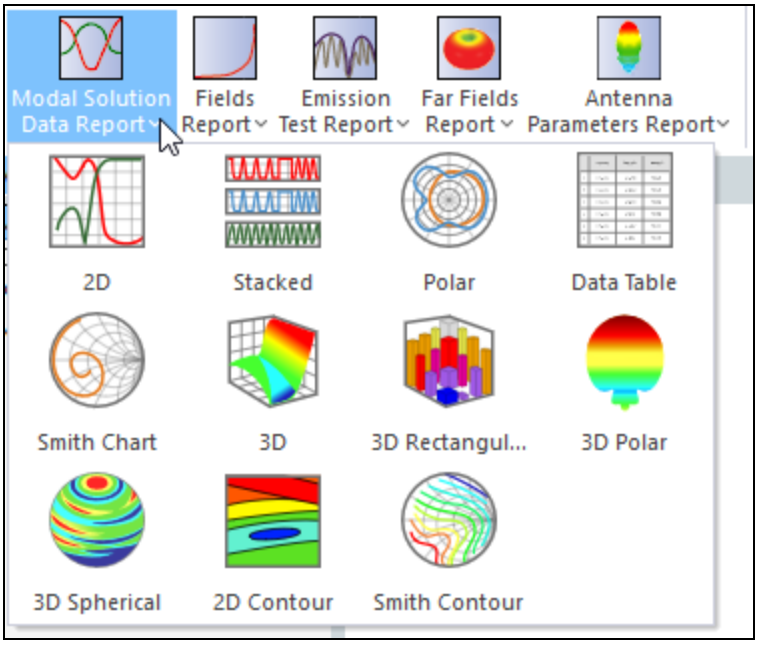
Creating Reports

After a solver generates a solution, you can analyze all the results for that solution. Ansys Electronics Desktop lets you create 2D or 3D plots. A 2D or 3D plot shows the relationship between a design's values and the corresponding results of the analysis. you can create reports using the **Create *<type>* Report** commands.

For each solution **<type>** (for instance, Modal, Terminal, Transient, Eigenmode, Characteristic mode, Fields, Far Fields, Emission test, Partial Discharge, Multipaction, and so forth), the **Results** menus present a list of **Create *<type>* Report** commands based on the solution data of direct interest for the design. For example, for the Eigenmode solution type, the **Results** menu contains templates for Eigenmode Parameters and for Fields. These appear on the menus as **Create Eigenmode Parameters Report** and **Create Fields Report**. For the Modal and Terminal Solution types, several different types appear, appropriate to each solution type. Each of these **Create *<type>* Report** menu items includes a further cascading menu that lists the Display Types available for that report. For some reports you can modify the Display Type from the Properties for that Report.

The **Results** tab for the Ribbon will show icons and drop-down menus for available report types for the active project.

Ansys Electromagnetics Suite 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 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 **HFSS 3D Layout** > **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 New Report

Following is the general procedure for creating a new report:

1. On the **HFSS 3D Layout** 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.

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 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 **HFSS 3D Layout** > **Results** > **Report Templates** > **PersonalLib** > *<templateName>*. You can also make such changes the default for new reports by right-clicking a modified report and selecting **Report Templates** > **Save Settings as default**.

When you have selected the report and display type from the **Results** menu, the **Report** dialog box appears, with the **Trace** tab selected by default.

2. In the **Y** Component section of the dialog box, make selections for the following:
   a. 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.

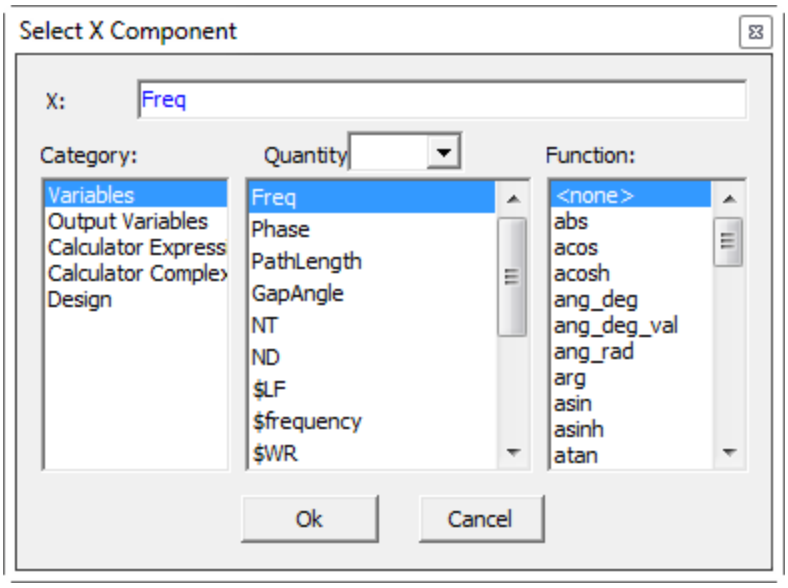
   b. **Function** list to apply to the Y quantities.
   c. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   d. **Range Function** button – opens the **Set Range Function** dialog box. This applies to the currently specified Quantity and Function.
   e. 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.
3. In the **X (Primary Sweep)** section, make selections for the following:
   a. Select the Primary value(s) 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.

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.

4. **Update Report** setting
   - **Real Time** checked – enable real time updates for all reports while the reports are being edited.
   - **Real Time** unchecked – enables drop down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.

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 **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.
6. 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.

7. 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

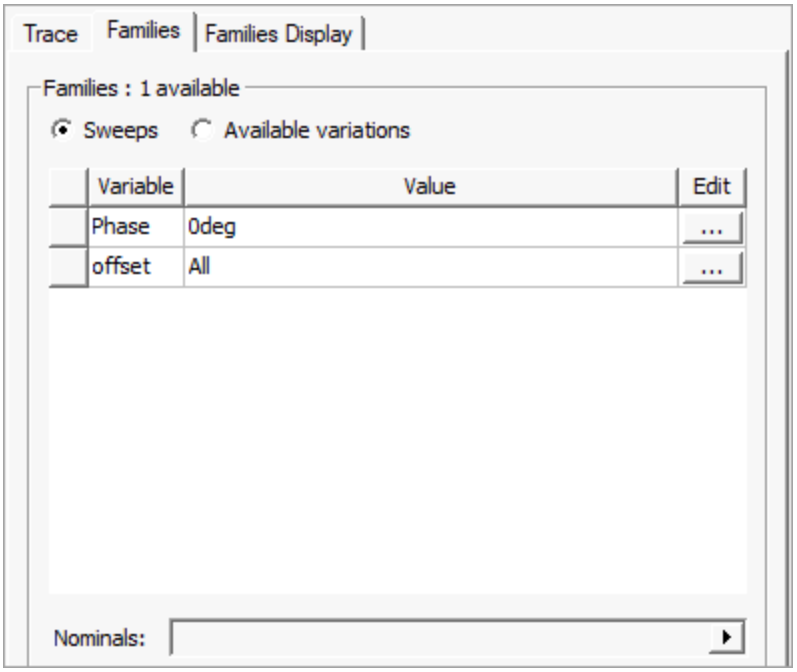
In the **Context** section, make selections from the following field or fields, depending on the design and solution type.

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.

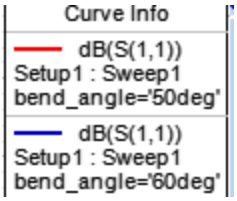
Using Families Tab for Reports

The **Families** tab of the **Report** dialog box provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined and solutions exist for multiple variable values (for example, for a parametric sweep or re-running an analysis with a different variable value). If no variables are defined, or none have solutions for different values, 0 families will be available. If so, the variables other than the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the solution value (which may be All, Nominal, or a Specific value), and an Edit column with an ellipsis [**...**] button. Families gives the number available. If an existing variable is specified as Nominal, only that value is currently available. You can set any solved variables as Nominal, All, or select from values provided for Available solutions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



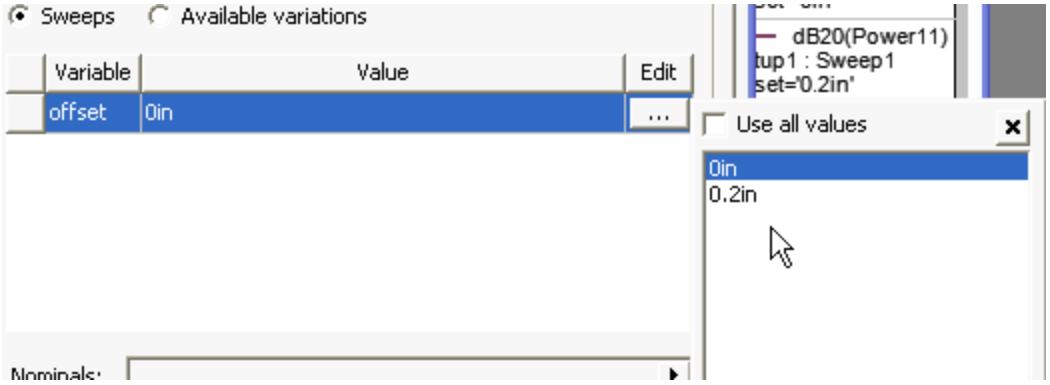
When you select a variable with multiple solved values, a trace for each solved value appears in the Report, with the variable value appended to the trace name in the Report legend.



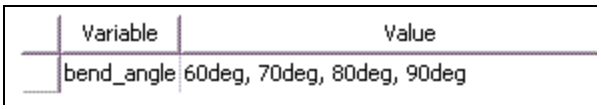
When families are available, you can make selections for the following:

1. Select the **Sweeps** radio button (the default) to list the swept variables you can select or the Available variations button to list and select variation values for which solutions exist.
2. With the **Sweeps** radio button selected, click the ellipsis [**...**] button to display a list of variable values for a particular variable. If many variables exist, you can use a scroll bar to navigate the list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

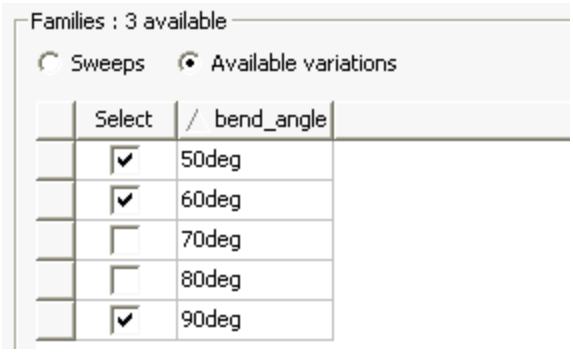


- To select all values, click the check box for **Use all** values. This writes "All" in the value field for that variable. You can also select individual values by clicking on them.
- To select a range of values, hold down the shift key, and click again.
- To select intermittent additional values, hold 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.

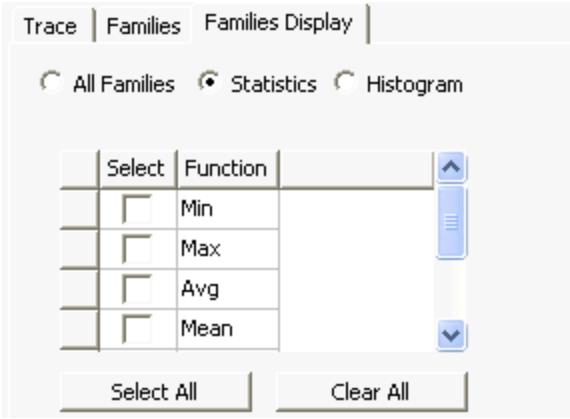


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

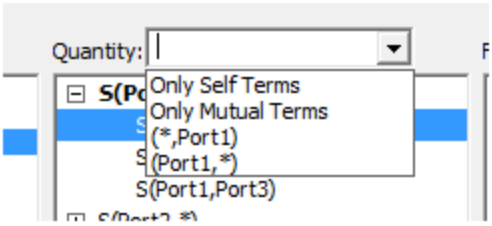


c. **Histogram** lets you select the number of bins to use for a histogram plot, and the sampling frequency to use.

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.

Deleting Reports

To use **Delete All Reports**:

1. Click **HFSS 3D Layout** > **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!

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.

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** – opens the **Output Variables** dialog box.
- **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
- **Update Trace** – updates the selected traces in a report based on further processing or changes.
- **New Report** – adds a report to the Project tree under the Results icon. The new Report is displayed in the main window.
- **Options** – opens the **Report Setup Options** dialog box. This contains a check box for using the advanced mode for editing and viewing trace components. This mode is automatic if the trace requires it. It also contains a field for setting the maximum number of significant digits to display for numerical quantities.
- **Close** – closes the **Report** dialog box.

The updated report appears in the view window.

4. **Update Report** setting
   - **Real Time** checked – enable real time updates for all reports while the reports are being edited.
   - **Real Time** unchecked – enables drop-down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.
5. In the **Context** section you make selections depending on the design and solution type.
6. The Families tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.
7. In the **Y** Component section of the dialog make selections for the following:
   a. Categories – those depend on the Solution type and the design. The selected category provides the default name of the plot, for instance S Parameter Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Quantities for Y are relative to the selected category.

> **Note:**
>
> The Quantity text field can be used to filter the Quantity list by typing in text, or by using the four predefined selections. This is useful if the Category selected produces a lengthy Quantities list. See Filtering Quantity Selections for the Reporter.
>
> When the matrix is very large, the number of quantities can be correspondingly huge. Therefore, the Quantities field can optionally use a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. See Report Setup Options.

c. **Function** list to apply to the Y quantities.

d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

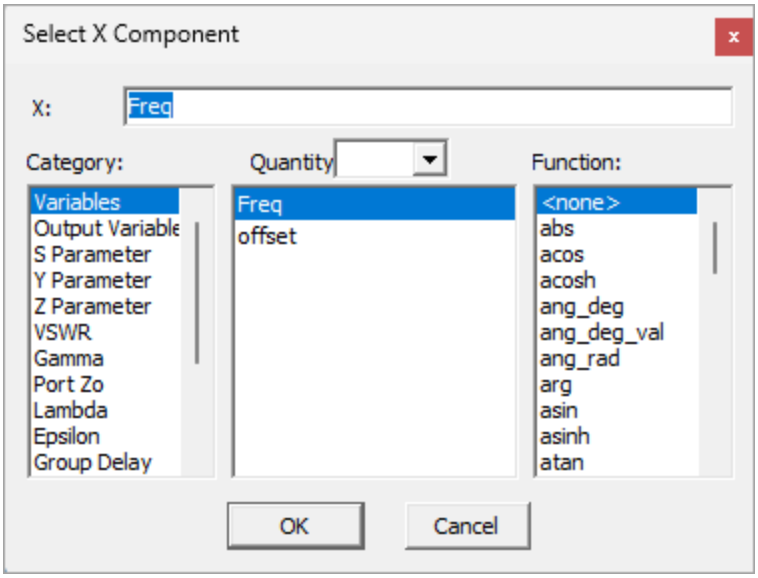
e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

8. In the **X (Primary Sweep)** section, make selections for the following:

a. Select the Primary value(s) from the drop-down menu.

To select an X component that is different from the Primary Sweep, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog to assign the X component.

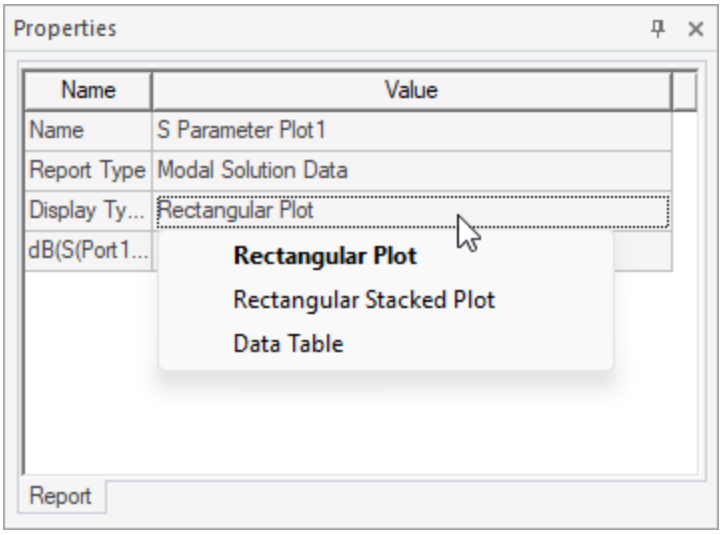
b. If sweeps are available, you can select **Browse** [...] to display a panel that lets you select Use all values, or selected sweep or sweeps, or access an Edit Sweep dialog box with further editing options. Post-Processing variables are Post-Processing sweeps/editable sweeps, so you can use the *Edit Sweep* dialog box to create your own sweep.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

You can also view and edit the properties of Reports and their traces via their Properties windows. See Modifying the Background Properties of a Report.

You can also modify the display type of an existing plot from the Properties dialog box for that plot. Select the Report icon in the Project Manager to display the **Properties** dialog box. Selecting the Display Type field displays a menu with selections available for that plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



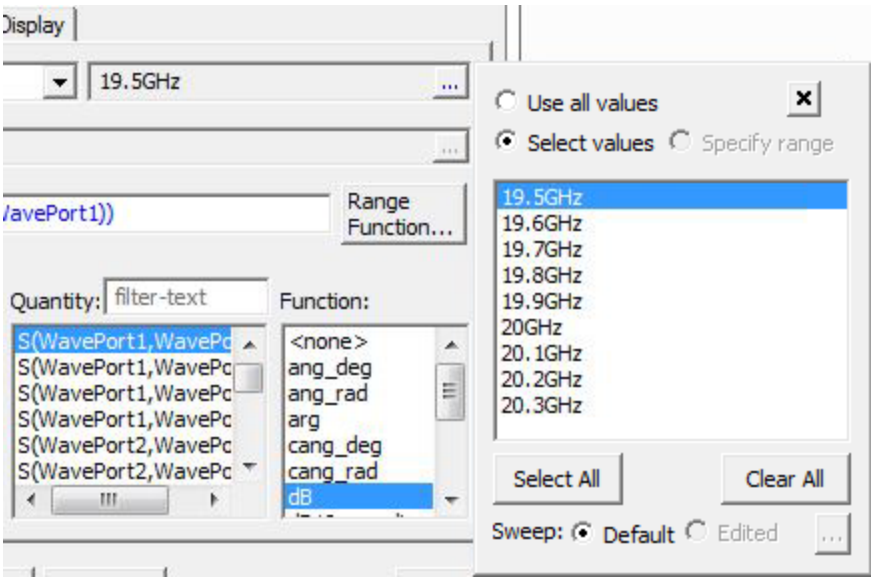
Once you make a selection, the plot display updates for the current selection.

> **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.

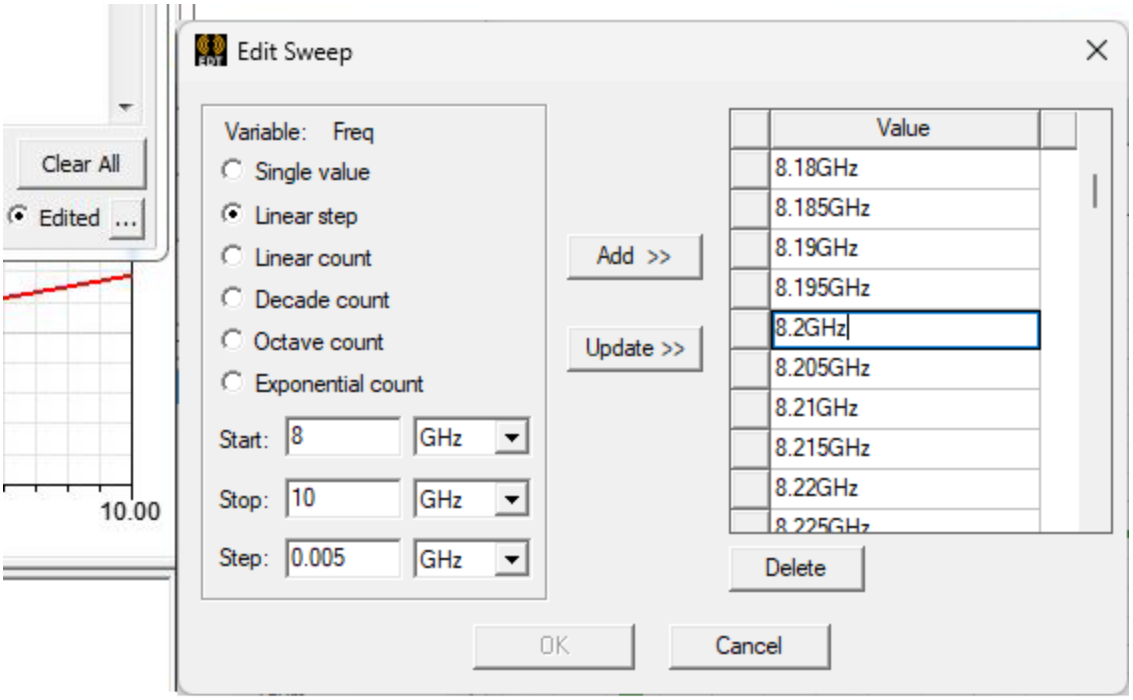
You select either the Sweep radio button for Default or Edited selection.

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.



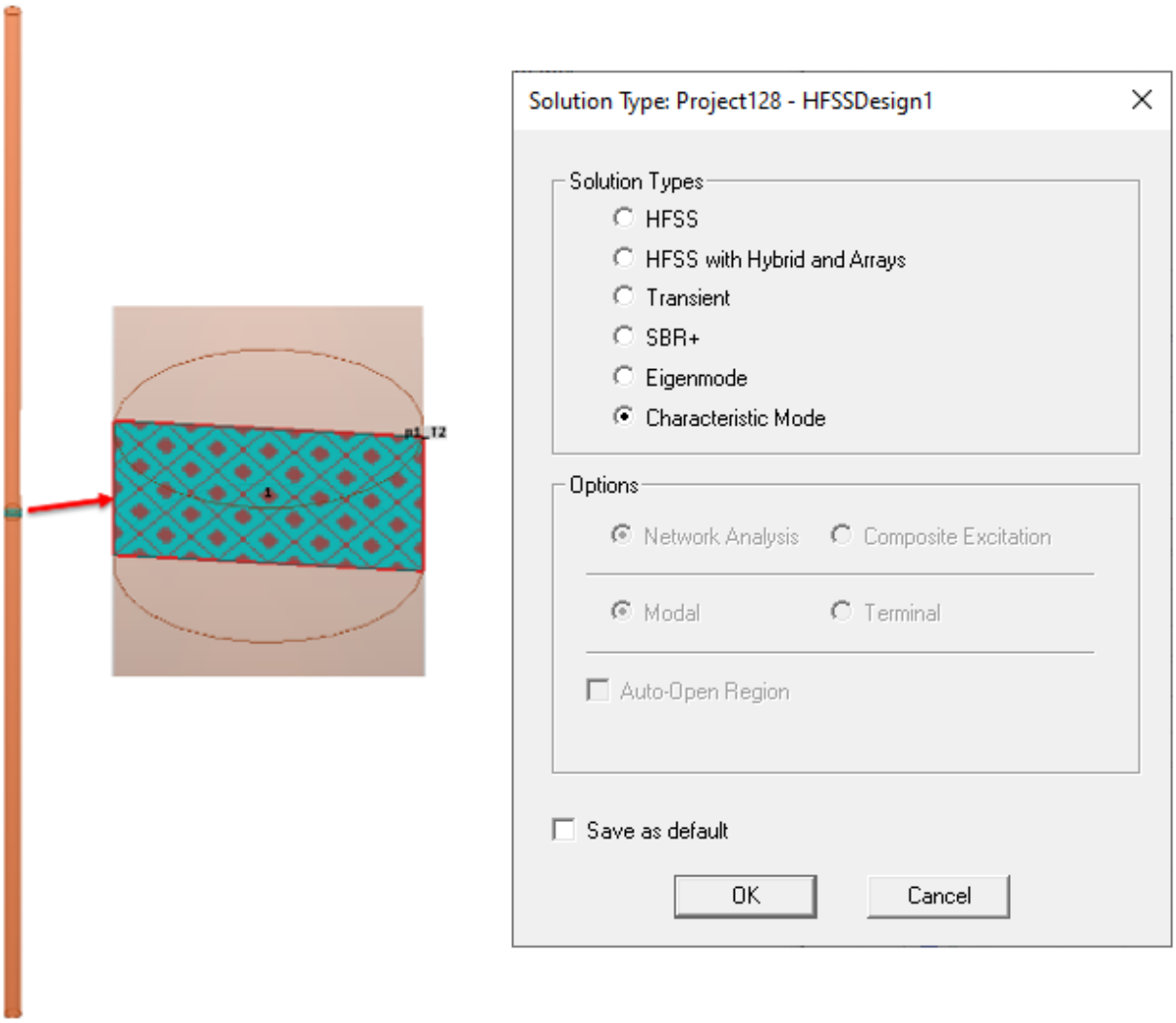
You can specify start, stop and step values and units, and add specific values to the list of current sweep values. The **Add>>** and **Update>>** buttons let you edit the value list.

You can use the mouse click, drag, and Alt+click to select values. You can also edit individual values.

Generating Reports For Characteristic Mode Analysis

This section describes how to use the characteristic mode analysis feature in HFSS when designing an antenna and generating characteristic mode analysis reports. The section also covers post processing capabilities available in HFSS when using CMA. To illustrate post processing for characteristic mode analysis, we will use a dipole antenna example. The two conducting arms of the dipole antenna are excited by a lumped port. From the menu item **HFSS > Solution Type**, define **Characteristic Mode** as the solution type.

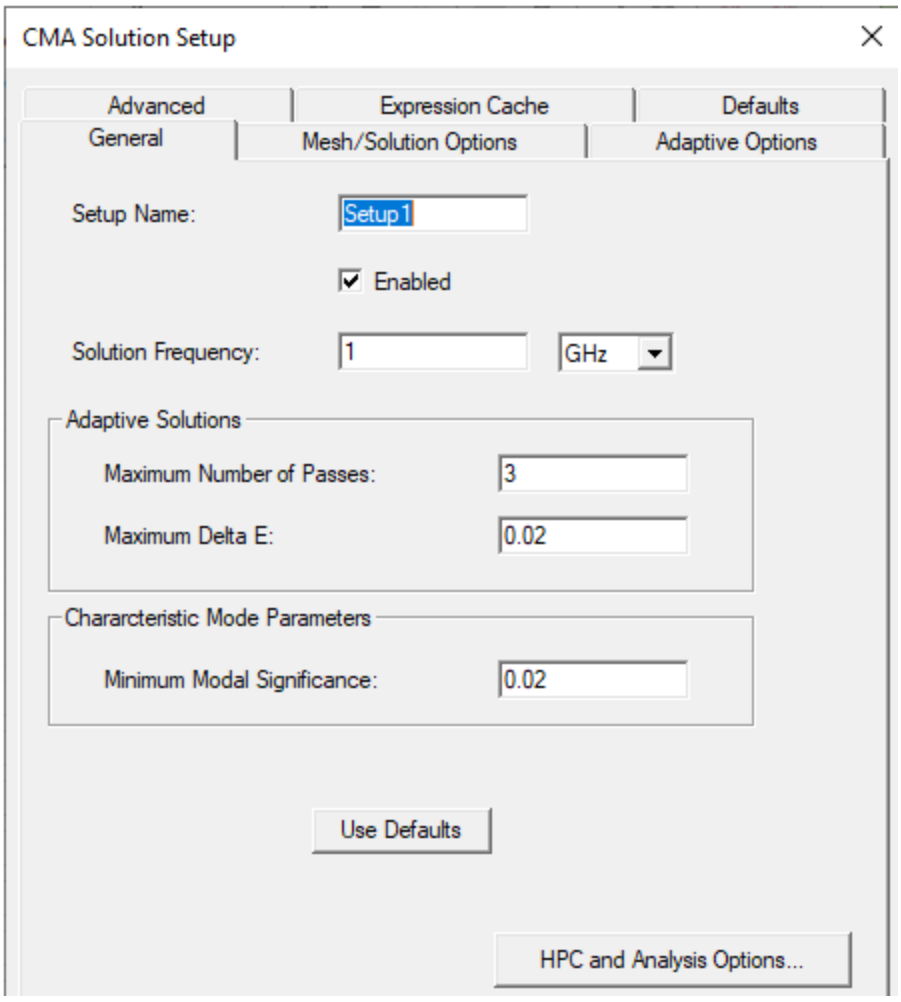
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Define the adapt frequency and characteristic mode parameters on the **CMA Solution Setup** dialog box. For Characteristic Modes solution types, these include the following:

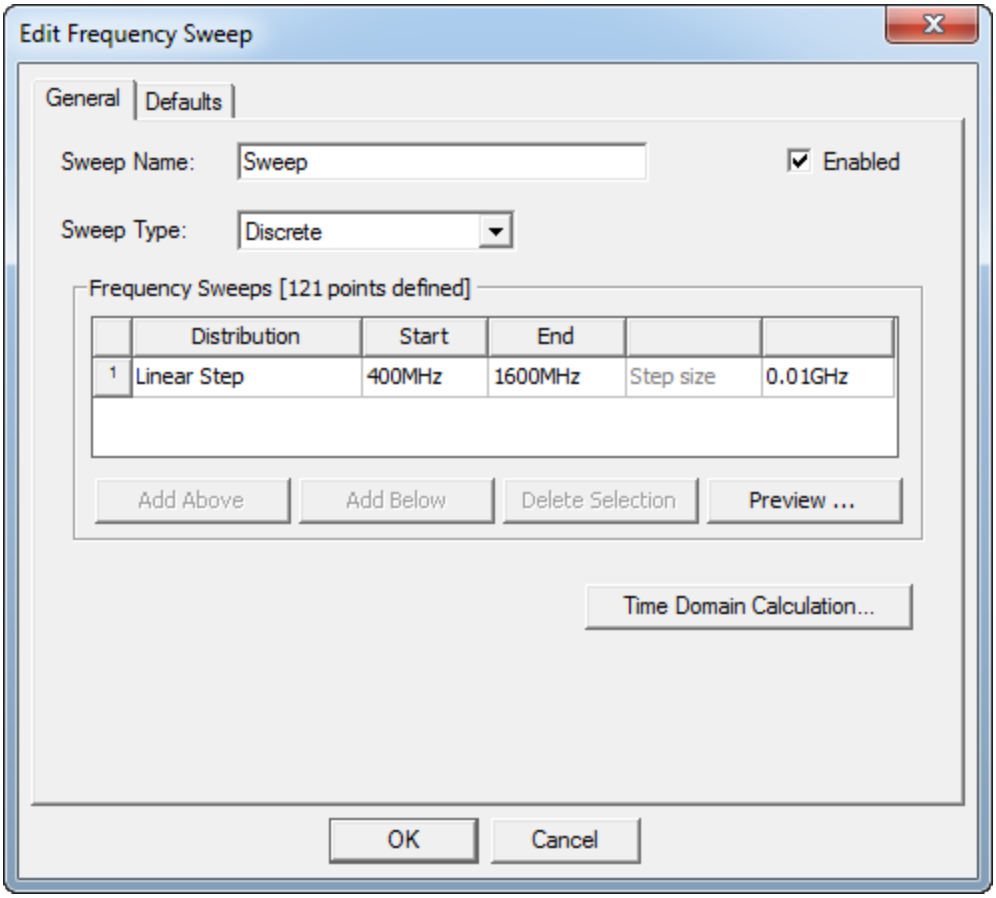
a. Solution Frequency
b. Maximum Number of Passes
c. Maximum Delta E
e. Minimum Modal Significance, a measure of how significant a mode is for solving the EFIE at a given frequency. It also determines how hard or how easy it is to excite a mode. The default for Minimum Modal Significance is 0.02. All the modes that are above this value 0.02 are calculated.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



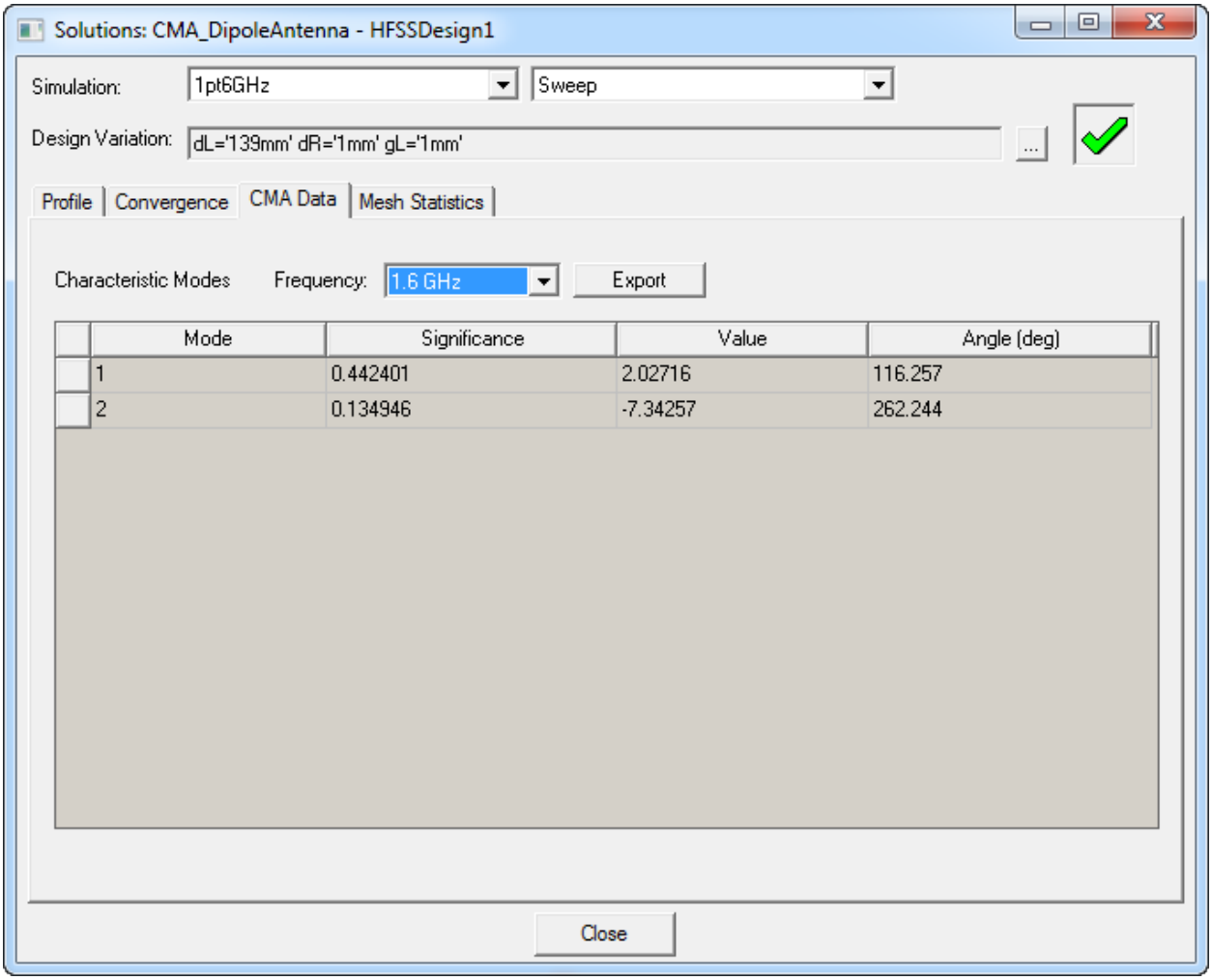
For this design the adapt frequency is set to 1.6 GHz. It's recommended that you define the adapt frequency to be the highest frequency in the discrete sweep.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After the simulation of the dipole antenna is complete, you can view the number of modes above the minimum modal significance, their characteristic angles, and characteristic values at every frequency in the **Solution Data** dialog box. To bring up this dialog box, right-click Results in the project tree and select **Solution Data** from the shortcut menu. The characteristic mode quantities are displayed in the **CMA Data** panel.

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 Frequency drop-down menu, select the desired frequency view all the quantities for the characteristic modes. As an example, at the adapt frequency of 1.6 GHz, there are two modes in this dipole antenna above the specified minimum modal significance.

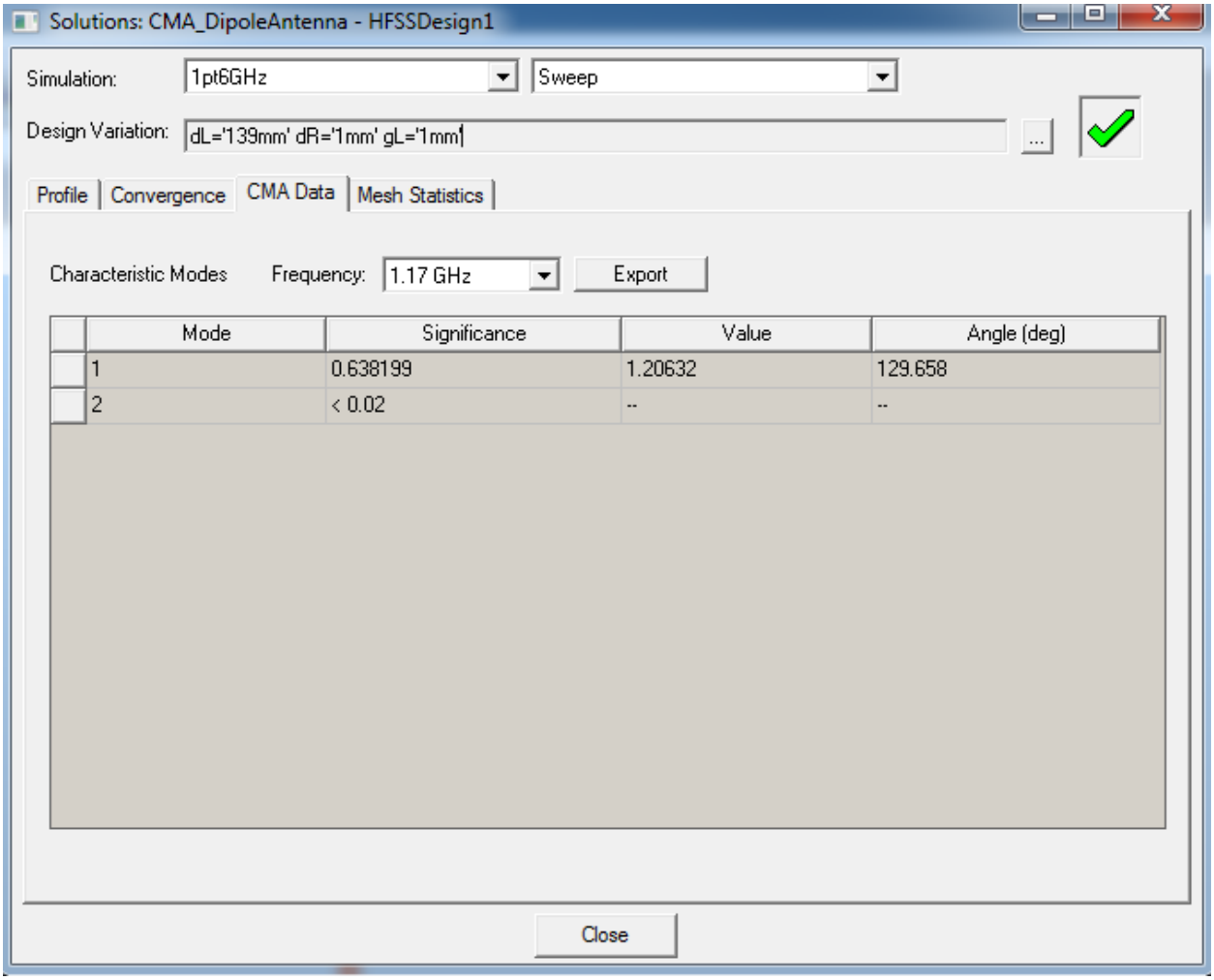
For these two modes:

- Modal Significance MS_1= 0.442401 and MS_2 = 0.134946 at 1.6 GHz
- Characteristic angles α_1= 116.257 degrees and α_2= 262.244 degrees.
- Characteristic values λ_1=+2.02716 and λ_2= -7.34257.

From these values, you can infer that mode 1 is inductive and mode 2 is capacitive.

As you choose lower frequencies, significance of mode 2 reduces and MS_2 approaches a value less than 0.02. At 1.17 GHz, only mode 1 is significant and above 0.02.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

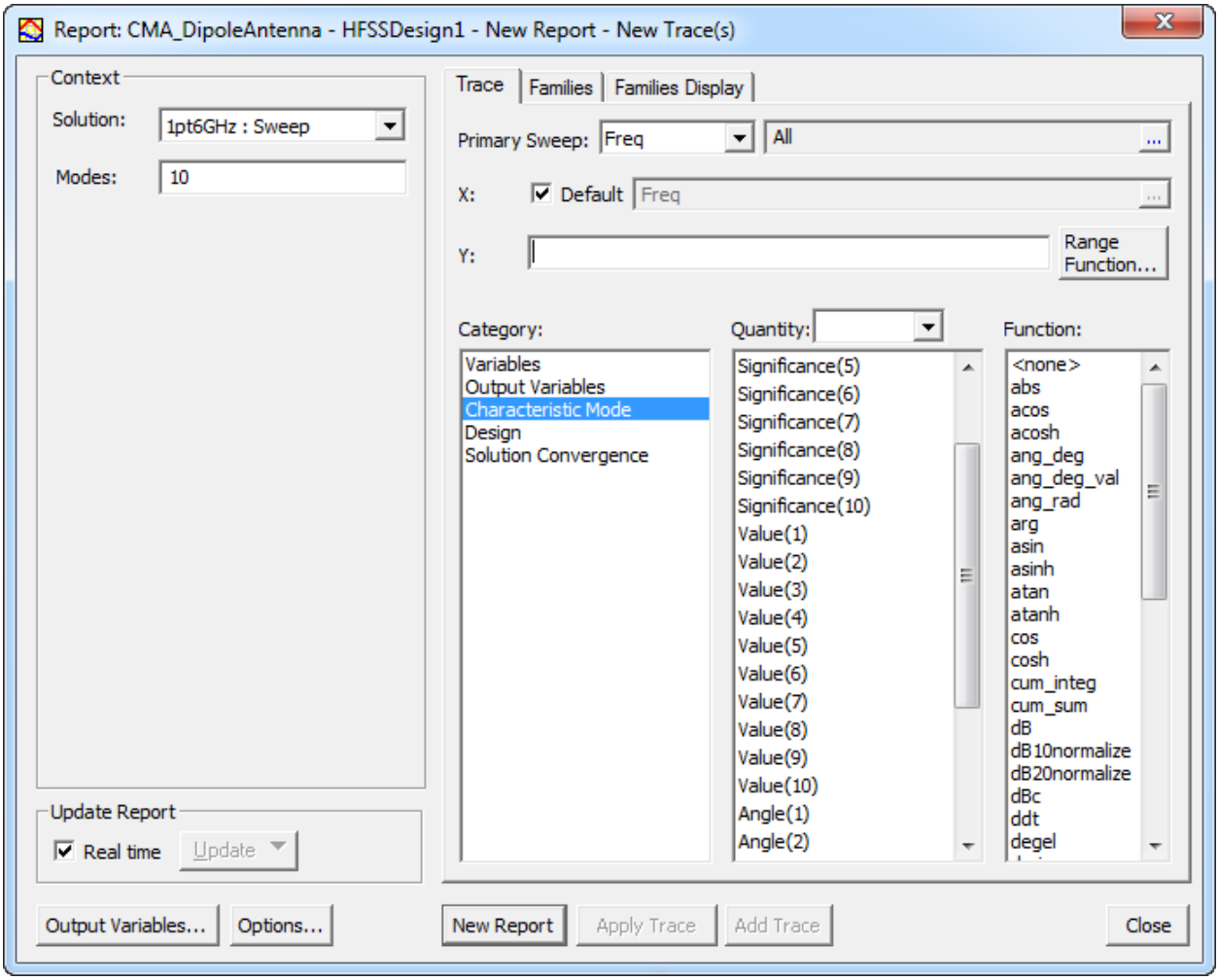


Characteristic Mode Data Report

The process for generating Characteristic Mode Plots for each of these quantities is as follows:

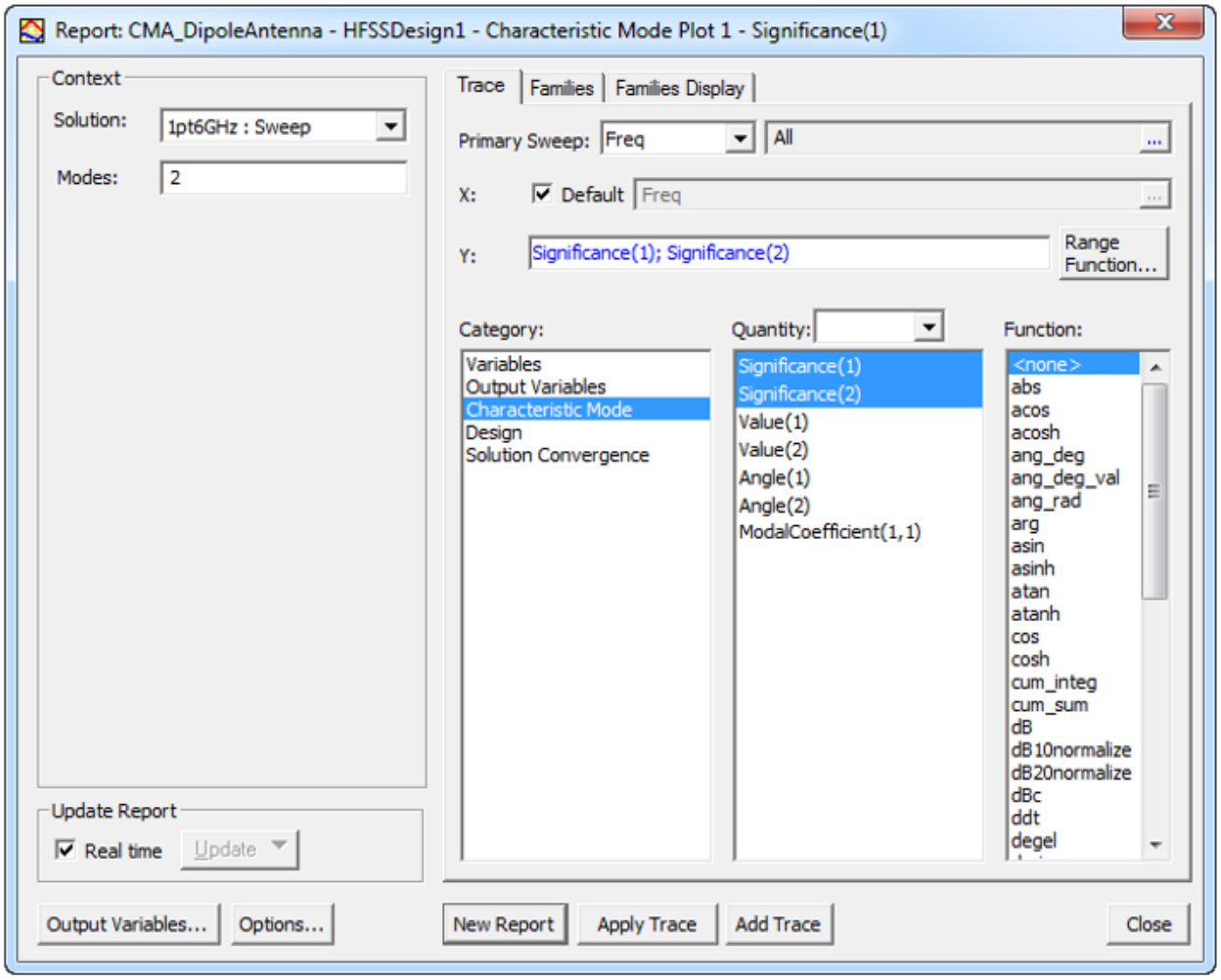
1. Check the maximum number of modes for a design from the **CMA Data** panel at the highest frequency.
2. Right-click **Results > Create Characteristic Mode Data Report > Rectangular Plot**, use the **Results** tab of the ribbon, and select from the **Characteristic Mode Data Report** menu and select the Rectangular Plot icon to open the **Report** dialog.

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 Context panel the number 10 in the Modes field is the default. You can either leave this value as is or change it to the maximum number of modes that are above the minimum modal significance. For this dipole antenna, the Modes field is set to 2 since there are only two modes at the adapt frequency as displayed in the CMA Data panel.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Ensure Characteristic Mode is selected in the **Category** and then select the **Quantity** of interest (Significances, Values, and Angles of the desired modes) and leave the **Function** as none.

   For CMA problems, you can also plot the Modal Coefficients. For Plotting Modal Coefficients, you must select a **Function**. These quantities are named "ModalCoefficient (<port>,<mode>) where <port> contains each of the names of the ports in the design and <mode> runs from the number of modes from 1 through N-1 modes where N is the number of modes specified by the user. The solution from a given port excitation is represented as a linear combination of the modal solutions.

$$J = \sum_{n} \alpha_n J_n$$

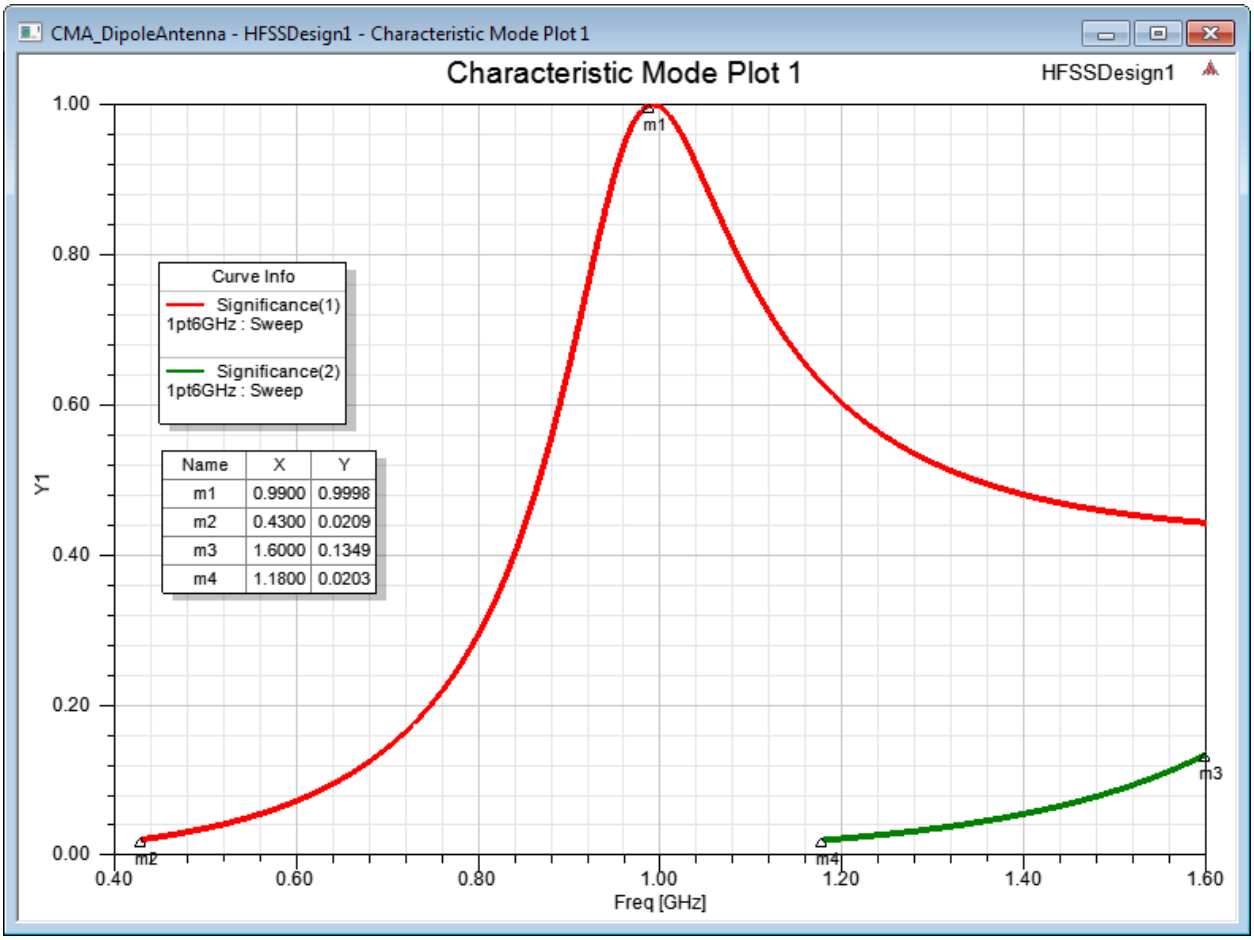
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The modal coefficients are the quantities which we multiply the modal solutions to obtain the port solution.

$$V_{n^i} = \langle J_n, E^i \rangle = \oiint_S Jn \cdot E^i ds.$$

The plots with their corresponding quantities of the characteristic mode are displayed here.

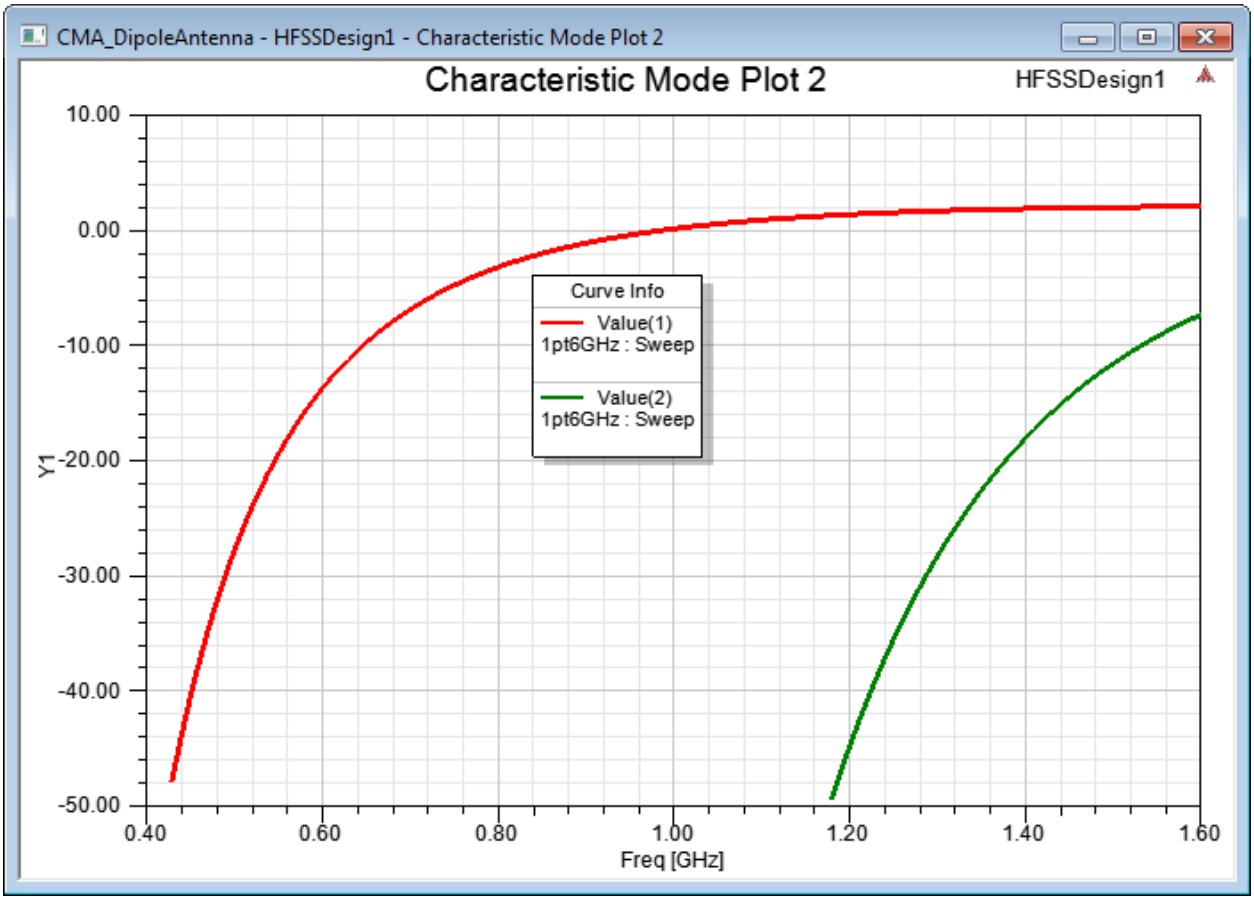
Modal Significance vs Frequency



For this antenna, mode 1 radiates the most at 0.99 GHz and its Significance is 0.9998. Mode 2 radiates most efficiently at 1.6 GHz and its maximum modal significance 0.1349.

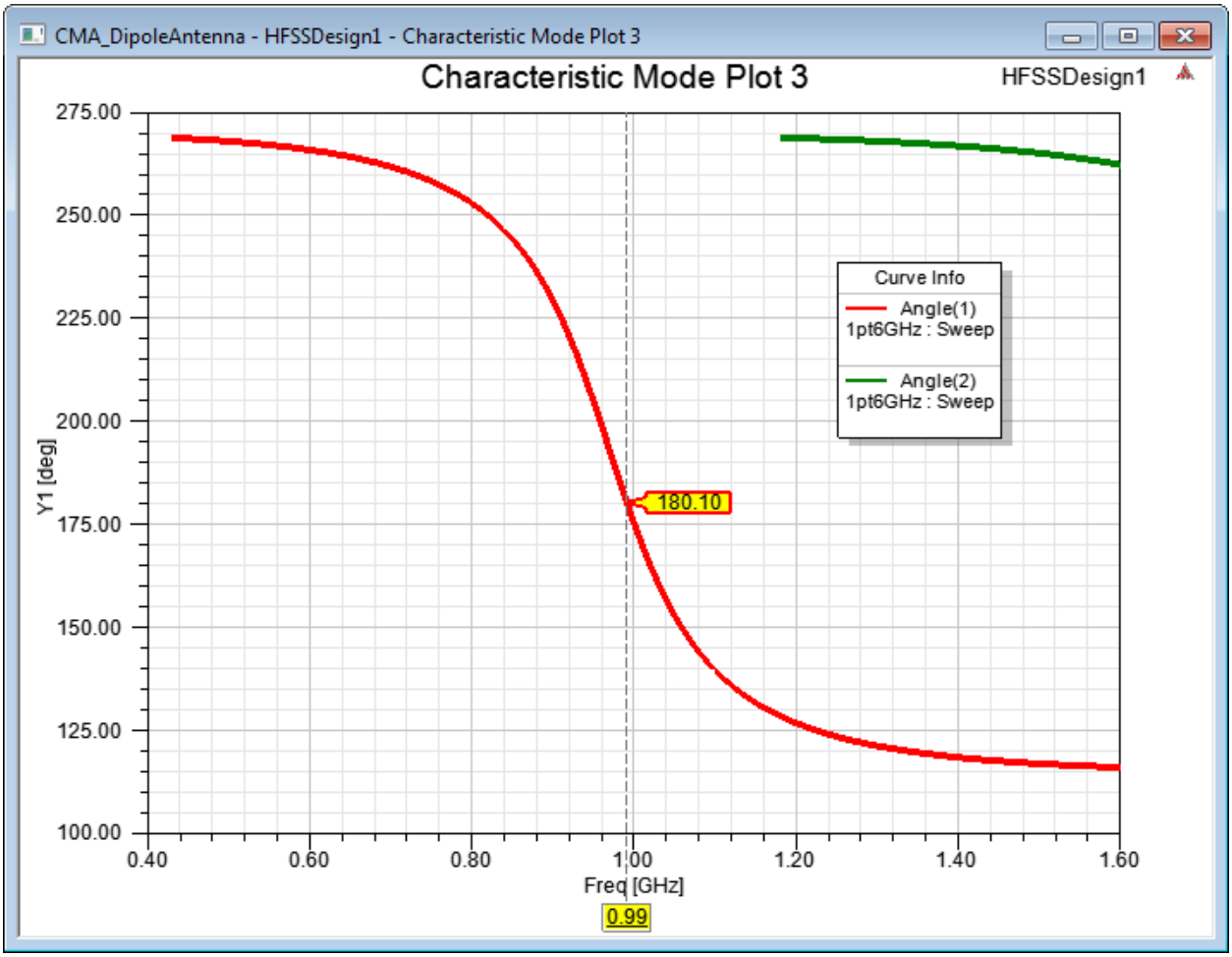
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Characteristic Value vs Frequency



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

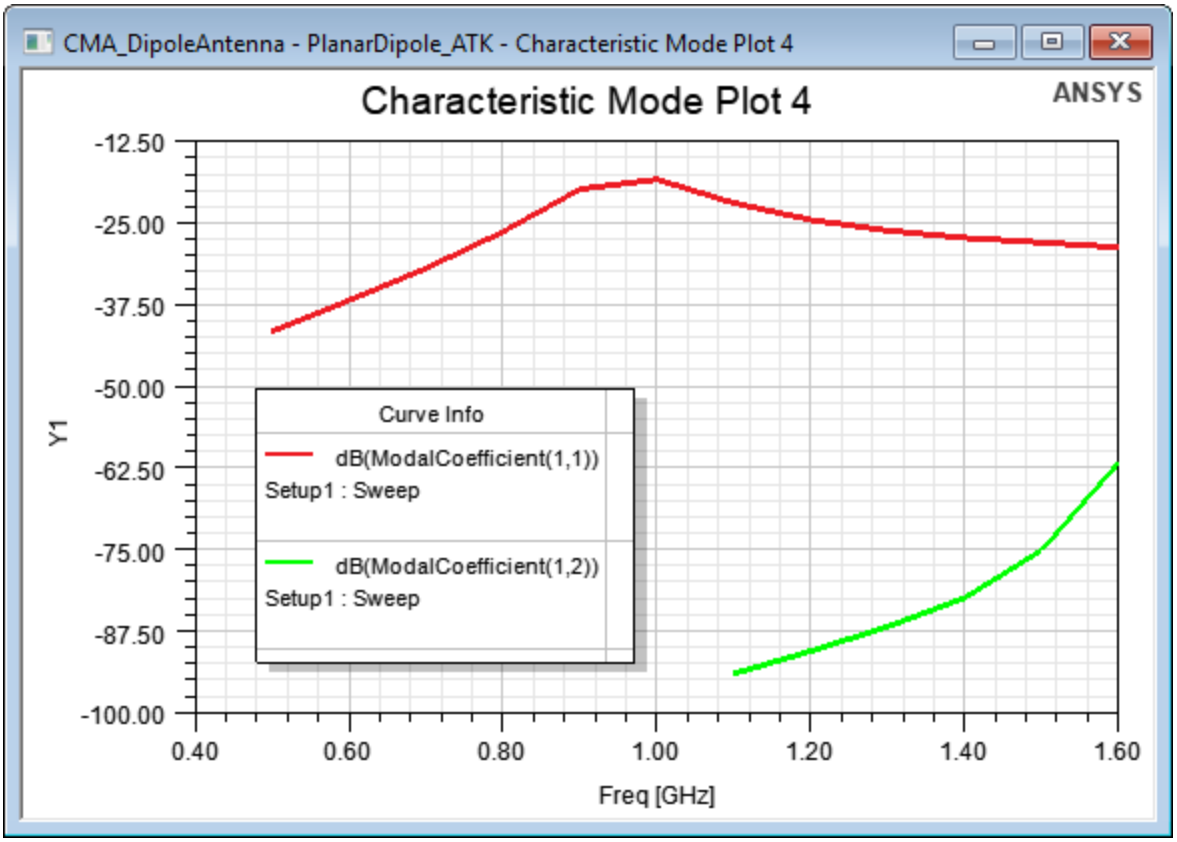
Characteristic Angle vs Frequency



The character angle for Mode 1 is 180 degrees, when it becomes resonant as shown in the above plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

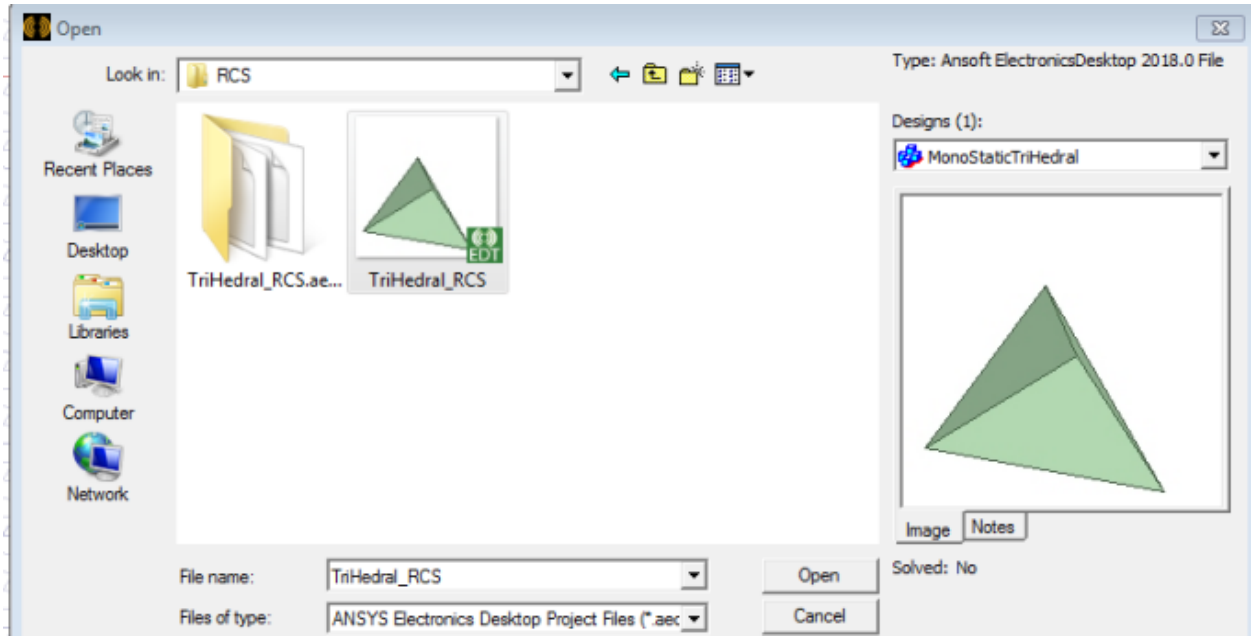
Modal Coefficient Plot



Generating Reports For Monostatic RCS

If you create a design contains both SBR+ Regions and Incident Plane wave you can then create reports for Monostatic RCS. The example projects for HFSS includes an example RCS project that demonstrates this.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

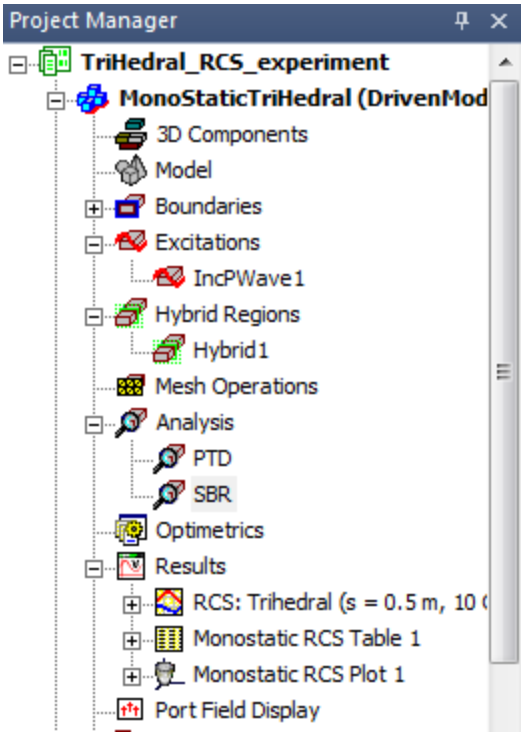


In the case of non-SBR+ projects:

- The new Monostatic report type works with non-SBR+ cases as well.
- For old version projects with Monostatic RCS report already defined it will still work but cannot be modified. The modify dialog will have far fields but not Monostatic RCS in the category column.

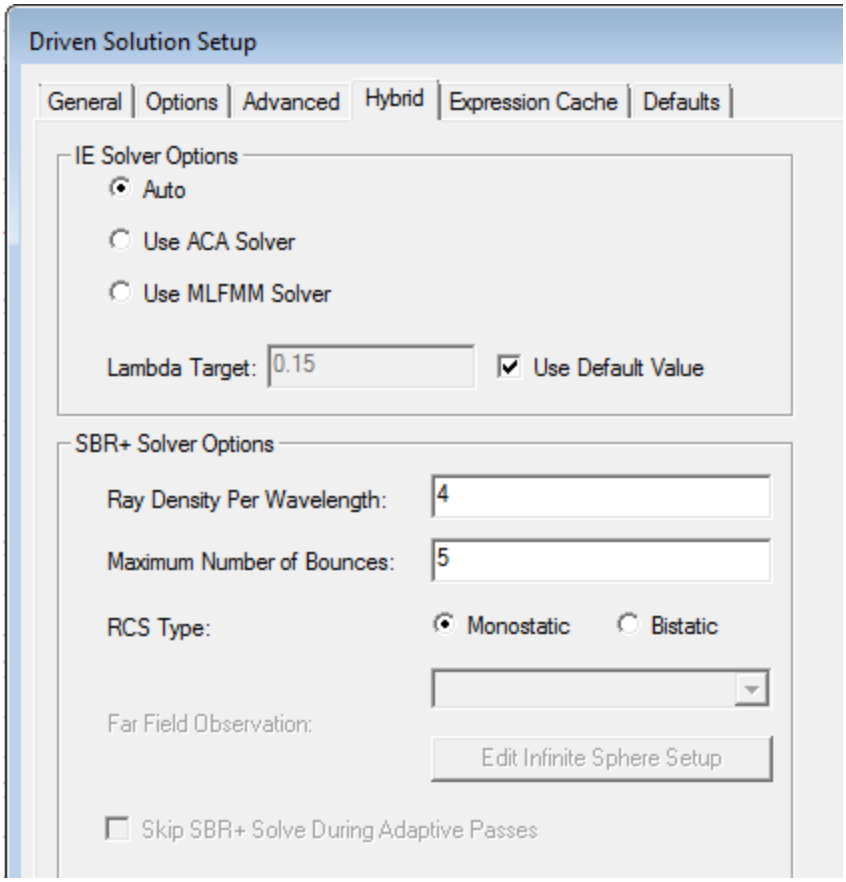
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The RCS example design includes an SBR+ Hybrid Region as well as an incident plane wave:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Designs with these elements include a selection for RCS Type as Monostatic or Bistatic.



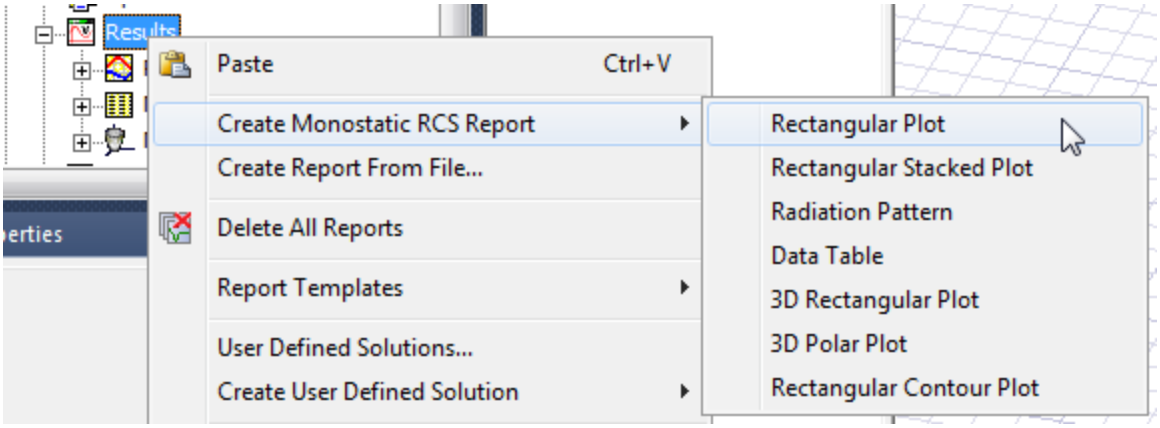
Selecting Monostatic permits you to create Monostatic Reports. In this case, the Infinite Sphere setup is disabled because the solver uses the direction of the Incident Plane wave to calculate the solution.

Monostatic RCS Reports

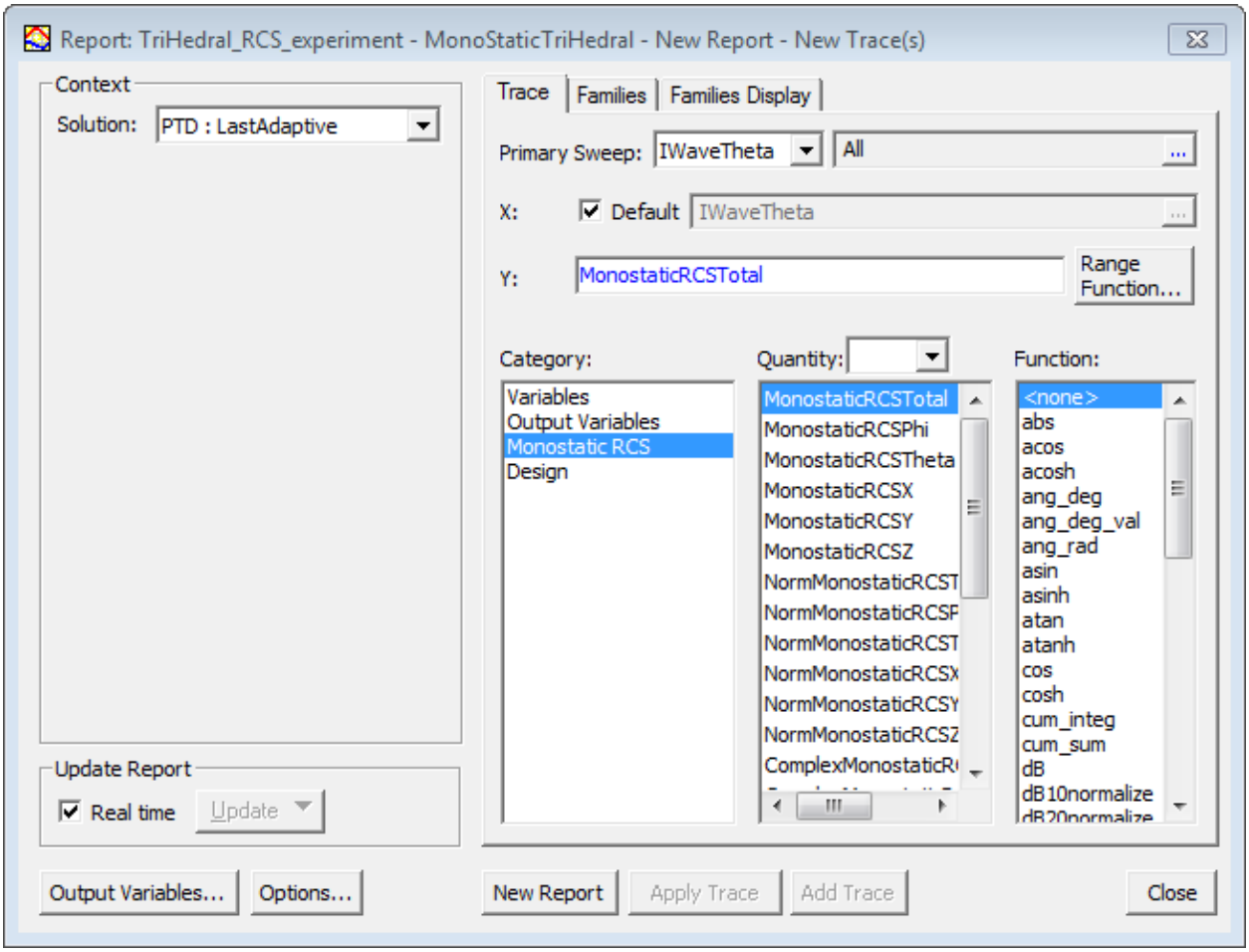
The process for generating Monostatic RCS Plots is as follows:

1. Solve a design containing SBR+ Hybrid Regions and an Incident Plane wave with no other excitations.
2. Right-click **Results > Create Characteristic Mode Data Report > Rectangular Plot** to open the **Report** dialog.

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 Context panel you can select the solution, but you do not need to select any geometry when using SBR+ and Monostatic RCS. Select the Category as Monostatic RCS to see a Quantity list for Monostatic traces you can create.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. You can choose between Freq, IWaveTheta and IWavePhi variables for specifying sweeps.

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 **HFSS 3D Layout** > **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.

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.

To zoom in using hotkeys:

- Press **Ctrl** + **"+"**

  If using the "=\+" key (not the "+" key on the numeric keypad) do **not** press the *Shift* 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 view zooms in 5 percent.

To zoom out using hotkeys:

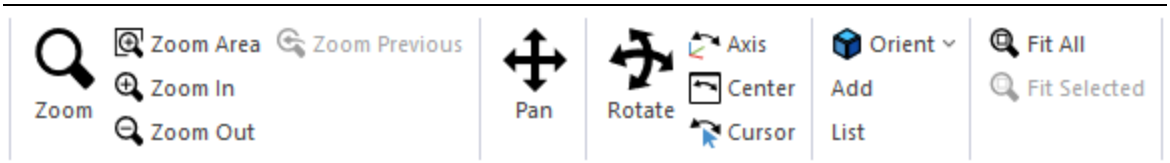
- Press **Ctrl** + **"-"**

  The view zooms out 5 percent.

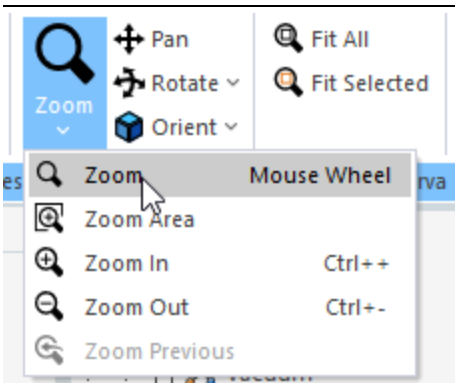
To zoom by dragging the mouse.

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.

   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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

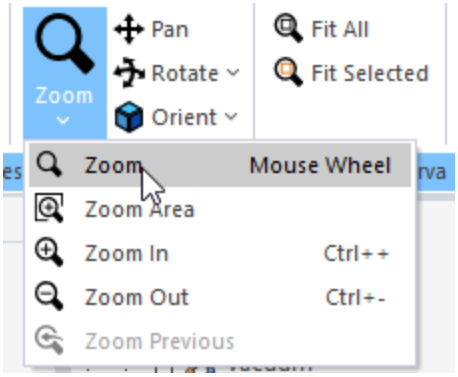
Zoom In and Zoom Out Commands

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.

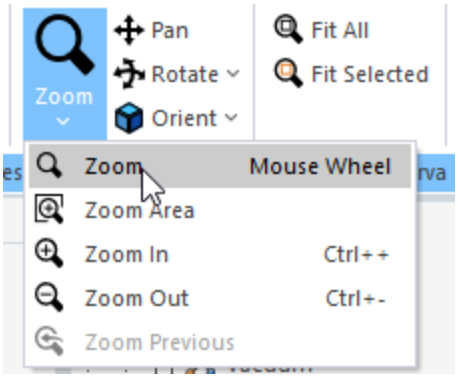
   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the **Zoom** icon, or press **Esc**.

Zoom Previous Command

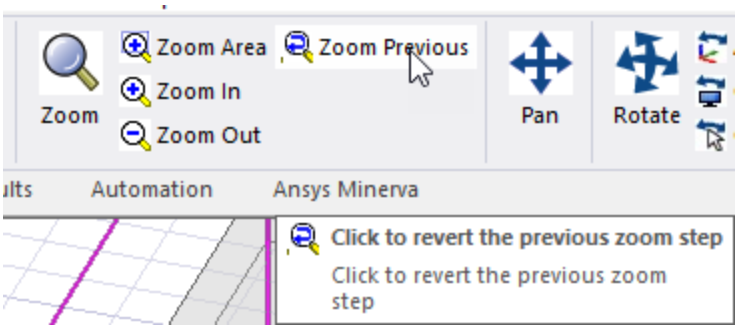
The **Zoom Previous** command is not enabled until at least one **Zoom** command has been used.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

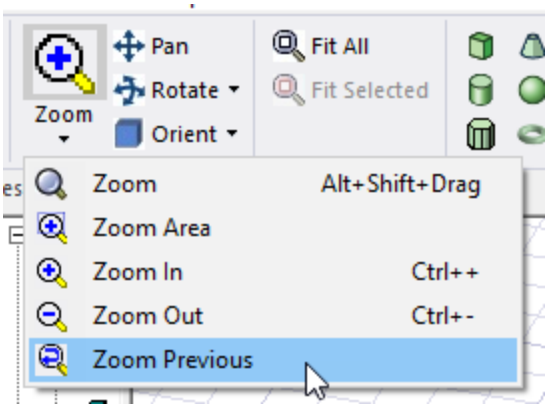


It operates on any existing sequence of **Zoom** commands. That is, it undoes the last previous **Zoom** command, and then, another exists, the one before that. To use **Zoom Previous**.

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**:



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.

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 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.

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**.

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
   - **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.
  - 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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 HFSS 3D Layout treats as zero, in the data, for example, dB Gain.

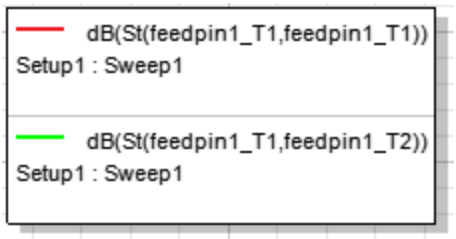
  - Map Infinity To – enter a real value for the Map Infinity Mode.

- **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:

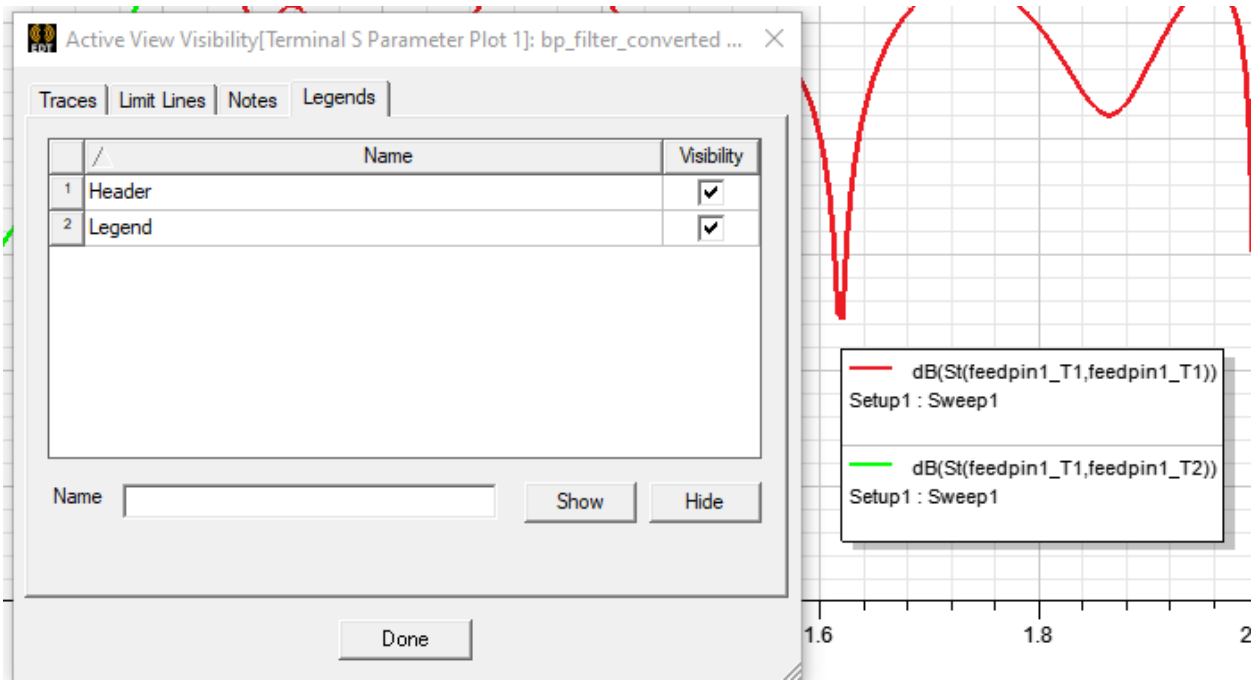
1. Make the report the active view.
2. Use **View** > **Active View Visibility** or the **Show/Hide** icons on the toolbar to display or hide the report.

   Either command displays the **Active View** dialog box.

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 **Legend** tab.

   This lists the legend (or legends) in the report.



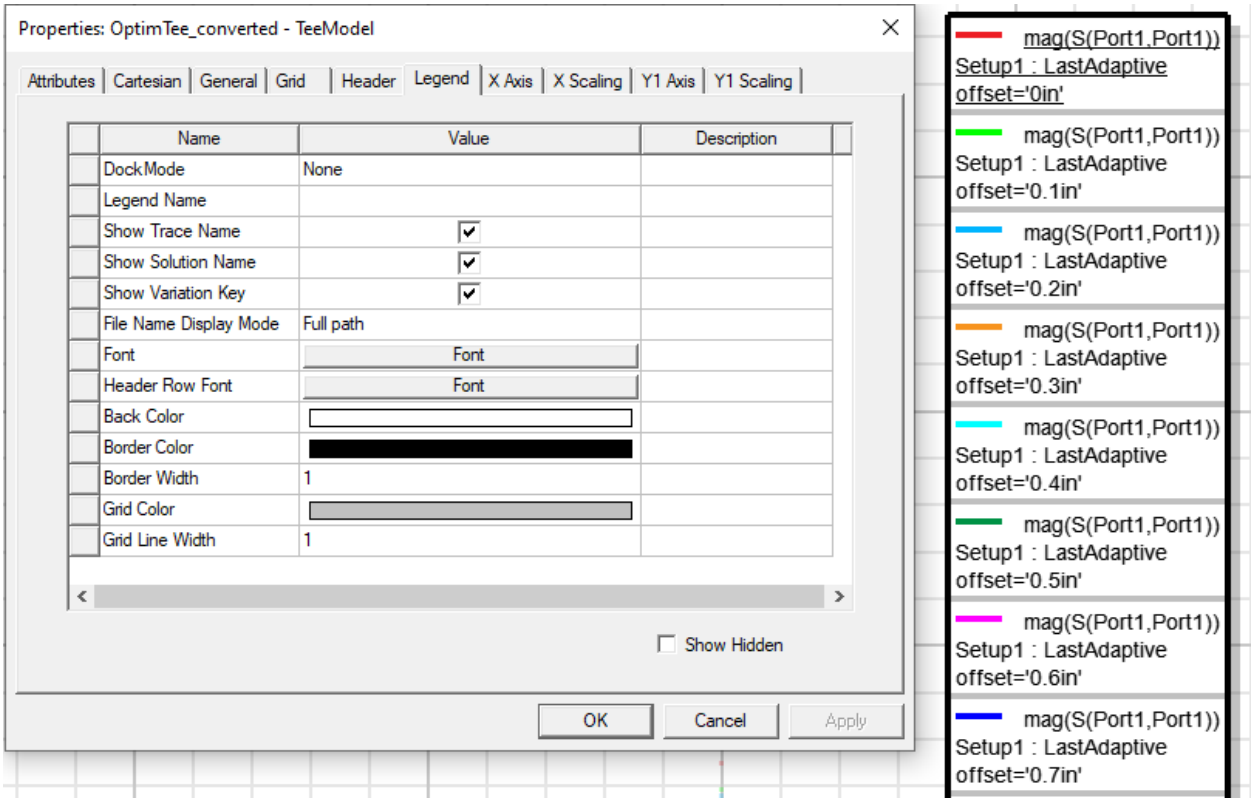
4. Enable the visibility check box, and click **OK** to close the dialog box and apply the change.

To edit the display properties of a legend:

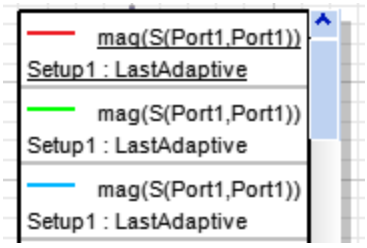
1. Double-click a legend in a report to display a docked properties window, or right-click on the legend and select **Edit** > **Properties** to display the floating **Properties** dialog box.

   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.

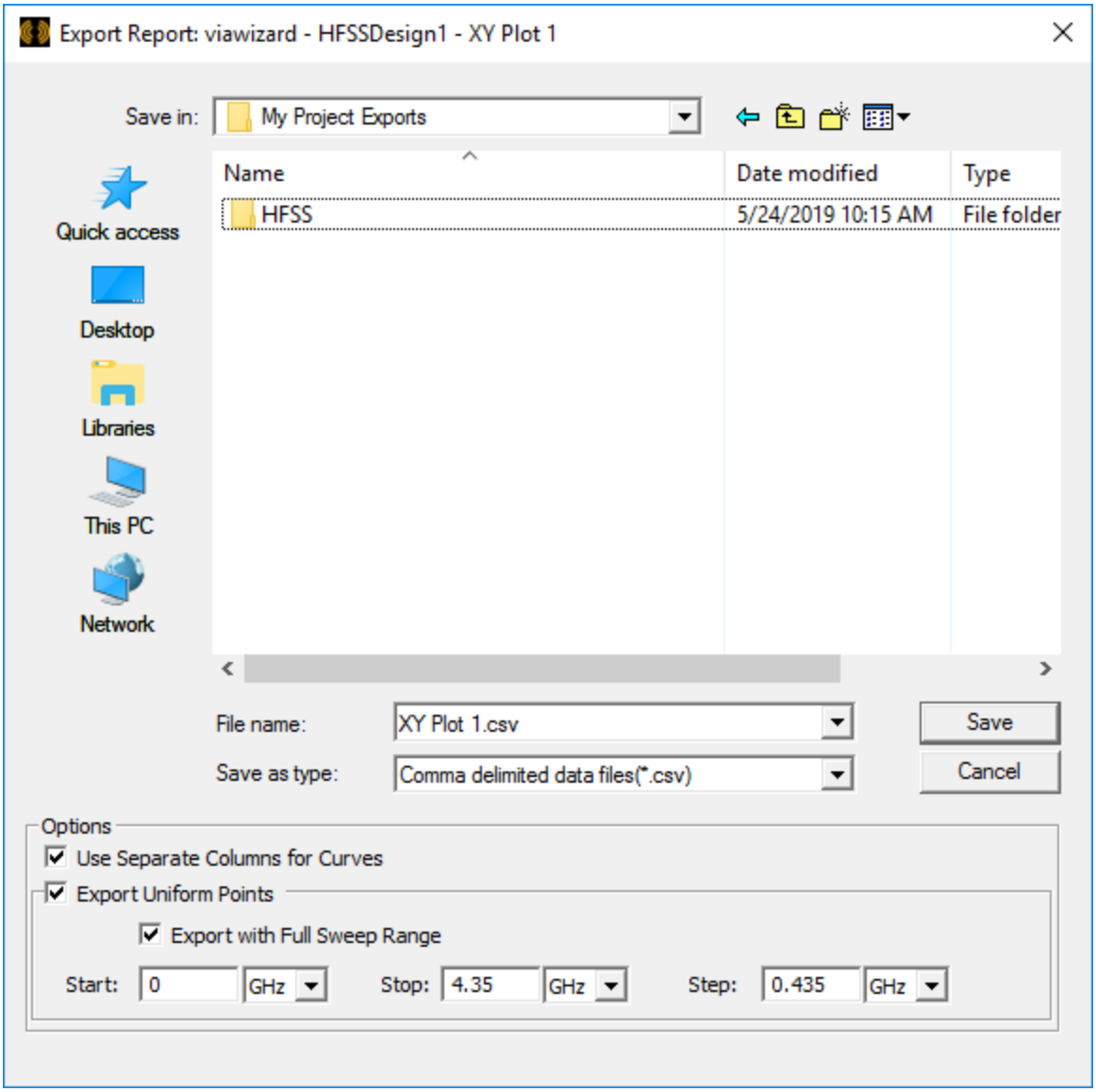
Exporting Ansoft Report Data Format Files

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.

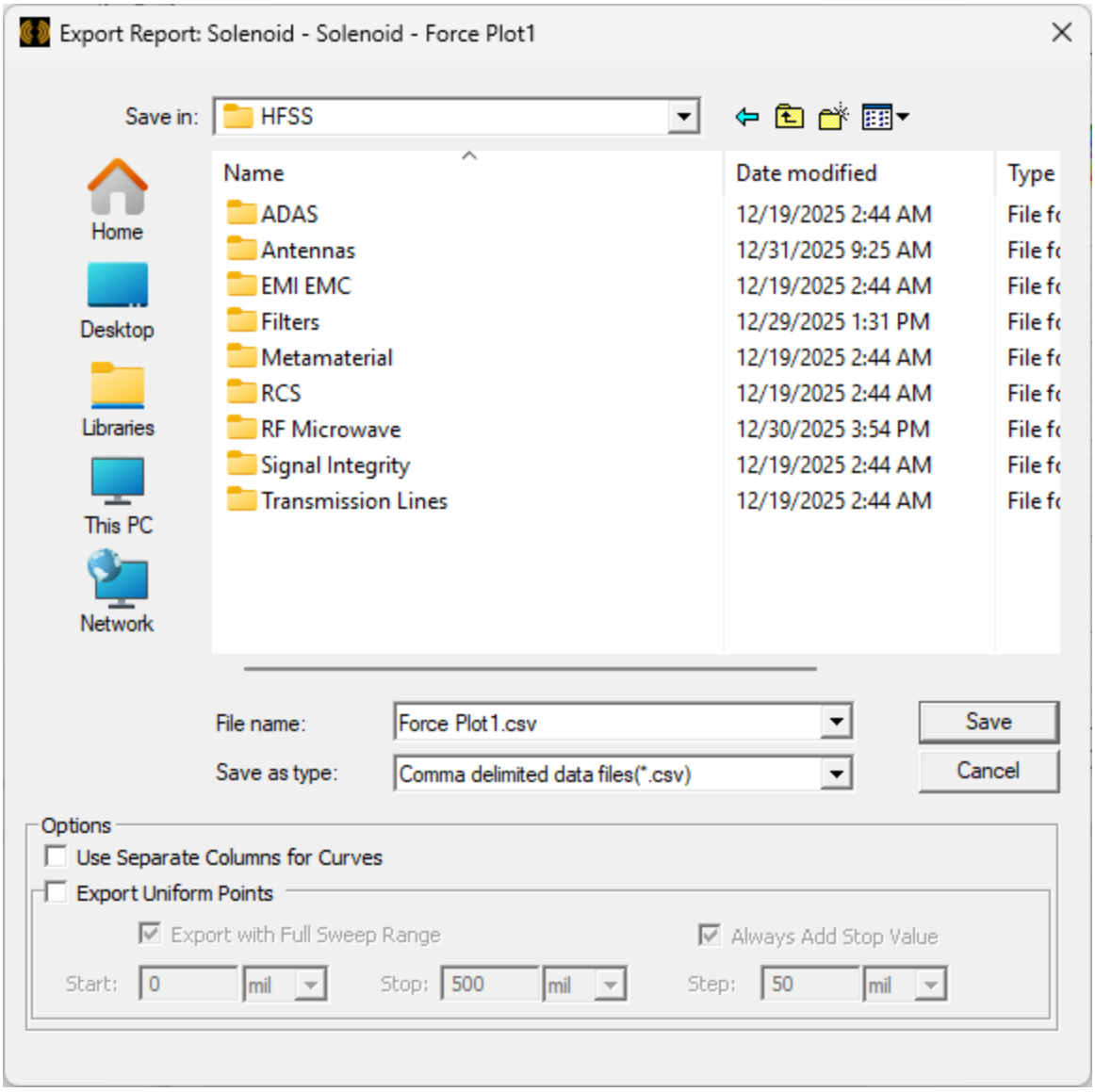
The file will then be available for import using Create Report from file.

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...**

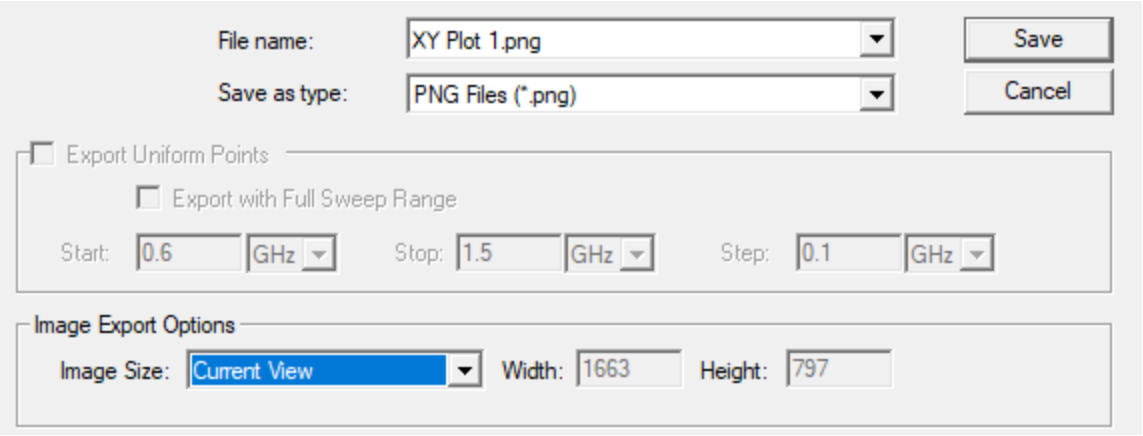
   The **Export Report** dialog box appears:



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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Contour** – for 3D plots, controls the appearance of the color map, including map type, ramp color, spectrum, IsoValType, levels, number of contours, and values shown.
- **Radiation Pattern** – for 2D polar plots, controls whether to show the circular grid and angle lines.
- **Stacked** – for stacked plots, controls properties for X scrollbar, thumb properties, and stack layout, auto fit, and stack height.
- **Smith** – for Smith charts, controls whether to show grids for Imp., Adm., Cir, and angle lines.
- **Traces** – controls the properties for traces, including: Color, Line Style, Line Width, Trace Type, whether to Show a symbol, Symbol Frequency, Symbol style, whether to Fill symbol, symbol color, and whether to show arrows. You can select traces either in the Legend or on the plot. See: Editing the Display Properties of Traces.
- **Axis** for X, Y or Z, or for Phi, Theta or Rho, and circular – the defaults for most of these values (applying to 2D and 3D both) are set in the **Report 2D Options**Axis tab.
  - Display name – check box for whether to display the axis name.
  - Specify name – check box for specifying the Axis name.
  - Name – this describes the axis to which the following properties/options refer. These are selected in the **Report** dialog box.
  - Axis Color – set the color by double-clicking to display the **Set color** dialog box. Select a default or custom color and click OK.
  - Axis Font – click the cell to display the **Edit Text Font** dialog box. The dialog box lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The dialog box also contains a preview field. OK the selections to apply the font edits and to close the dialog box.
  - Display Units – this specifies whether to display the axis units.
  - **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify Min – check box
- Min – text entry in same units as axis units. Saved as SI internally.
- Specify Max – check box
- Max – text entry in same units as axis units. Saved as SI internally.
- Specify Spacing – check box
- Spacing – text entry in same units as axis units. Saved as SI internally
- **Manual Units** (section):
  - Auto Units – use the check box compute the correct units for the axis.
  - Units – click on the cell to select from a menu of available units if you have not checked Auto Units.
- **Infinity Visualization** (section):
  - Map Infinity Mode – check box.

    Each axis can be set to treat infinity values in a user defined way. When you check the Map Infinity Mode, any infinity values in the input data get the infinityMap value (negative infinity get the value*-1 and positive infinity the positive value specified). This can be useful if there are zeros, or very small values that HFSS 3D Layout treats as zero, in the data, for example, dB Gain.

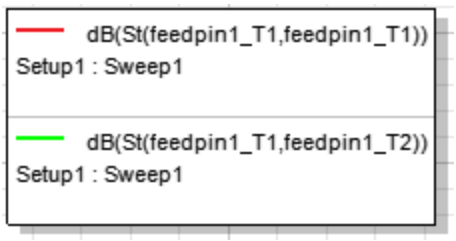
  - Map Infinity To – enter a real value for the Map Infinity Mode.

- **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:

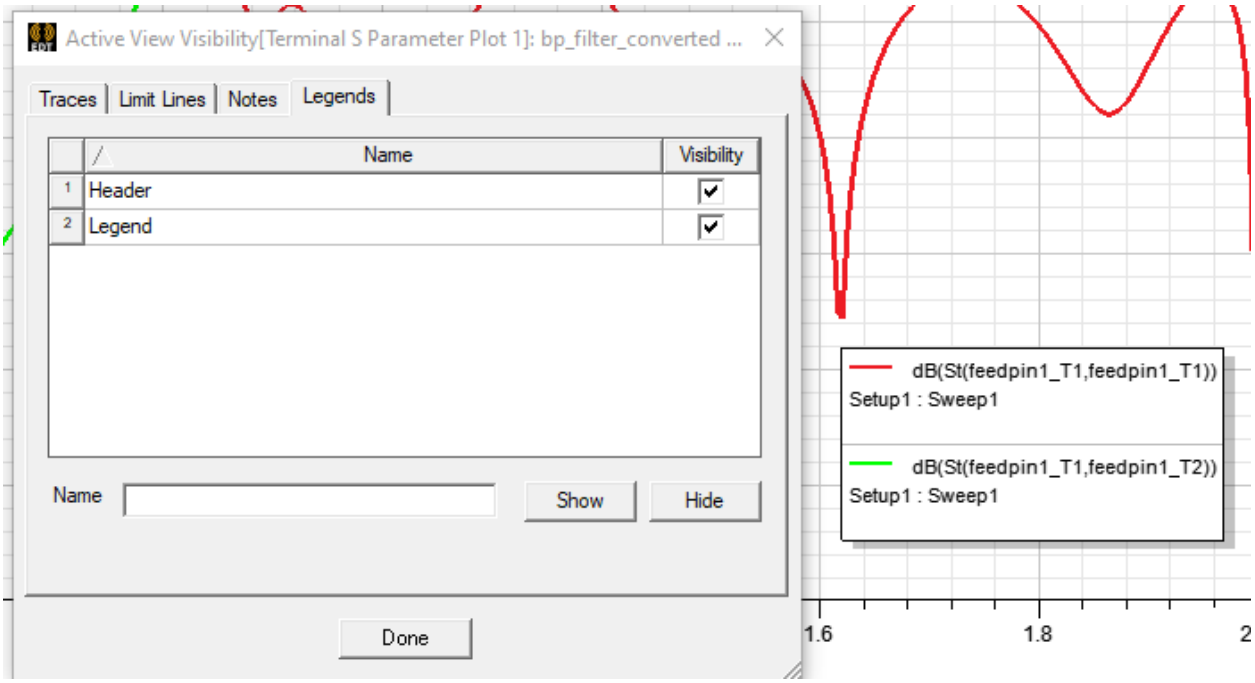
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



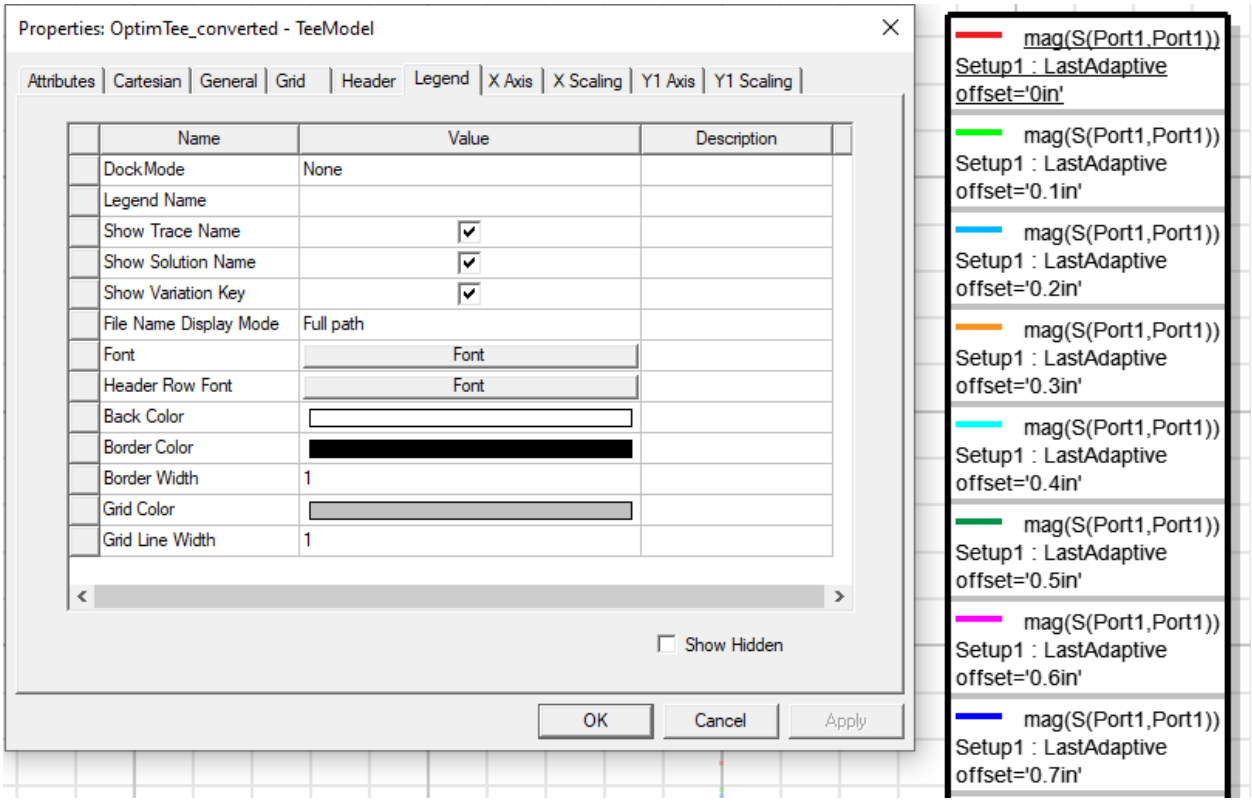
4. Enable the visibility check box, and click **OK** to close the dialog box and apply the change.

To edit the display properties of a legend:

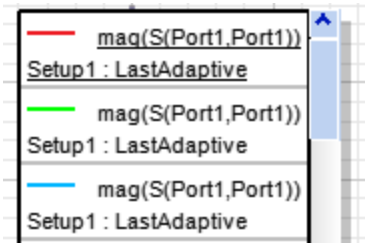
1. Double-click a legend in a report to display a docked properties window, or right-click on the legend and select **Edit** > **Properties** to display the floating **Properties** dialog box.

   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**.

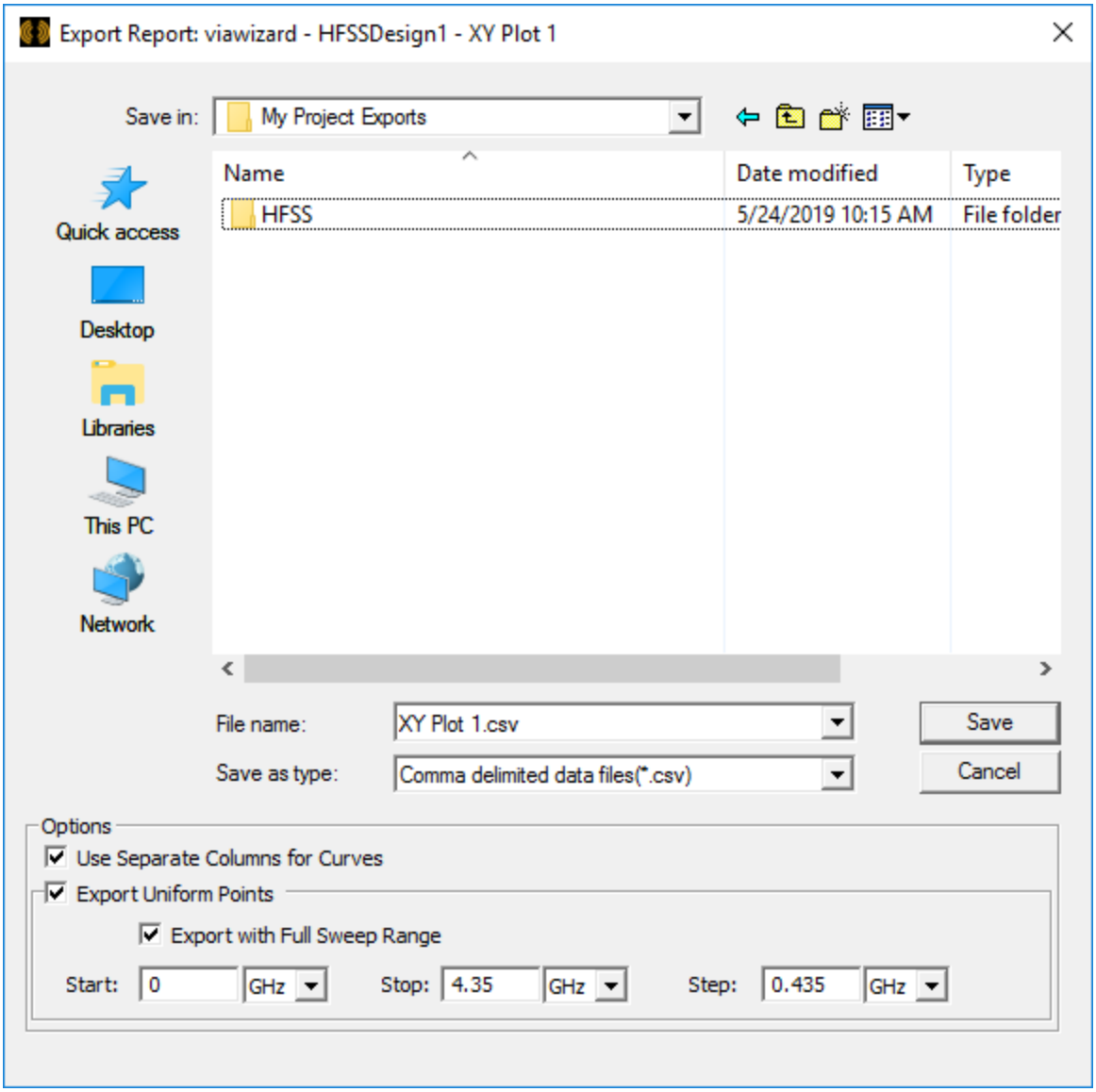
Exporting Ansoft Report Data Format Files

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.

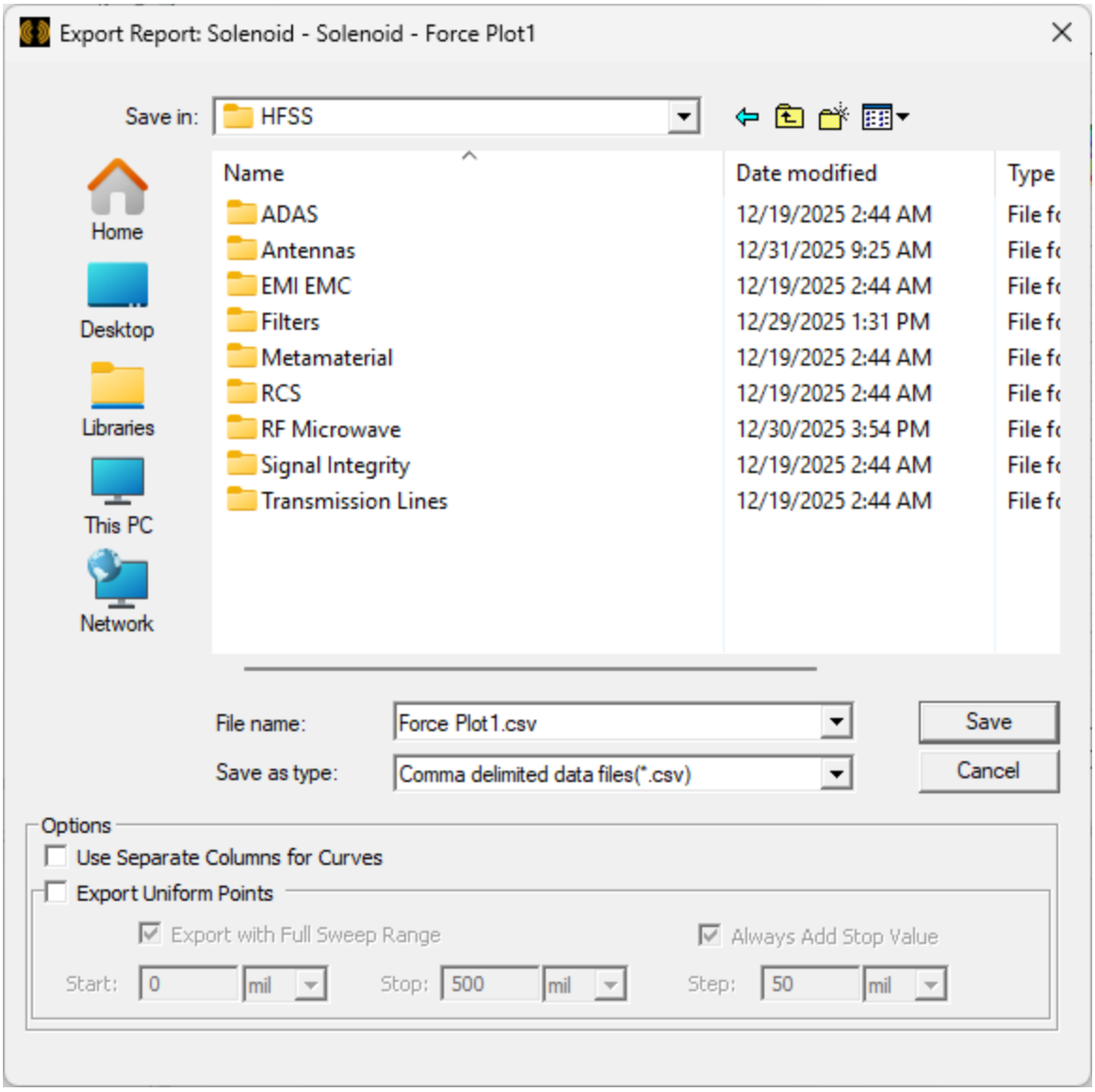
The file will then be available for import using Create Report from file.

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...**

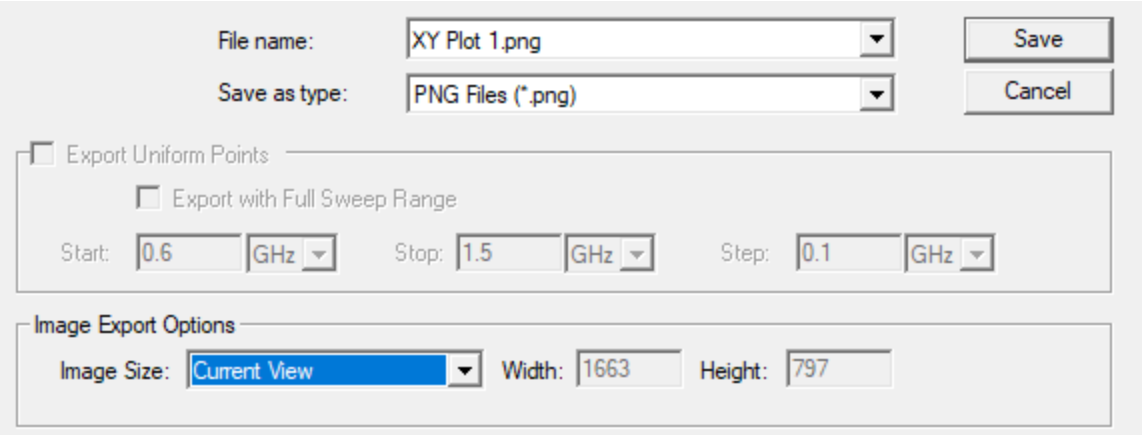
   The **Export Report** dialog box appears:



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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(such as for company name, or color scheme). Once, saved, this name will appear on the *PersonalLib* menu.

The **Save As Type** field currently supports the Ansoft Report Format (*rpt) format.

4. Click **Save** to save the template to the PersonalLib menu.

All *.rpt templates in the userLib directory appear on the **Results** > **Report Templates** > **PersonalLib** menu. Selecting a report from the PersonalLib menu opens a report that you can then Modify to add traces or perform other edits. Templates in the SysLib directory appear on the **Report Templates** menu.

To save an edited report as a default:

1. In the Project Tree, right-click the report name of interest to display the shortcut menu and click **Report Templates** > **Save Settings as default**. You can also click **Report2D** > **Report templates** > **Save Settings as default** or **Report3D** > **Save Settings as default**.

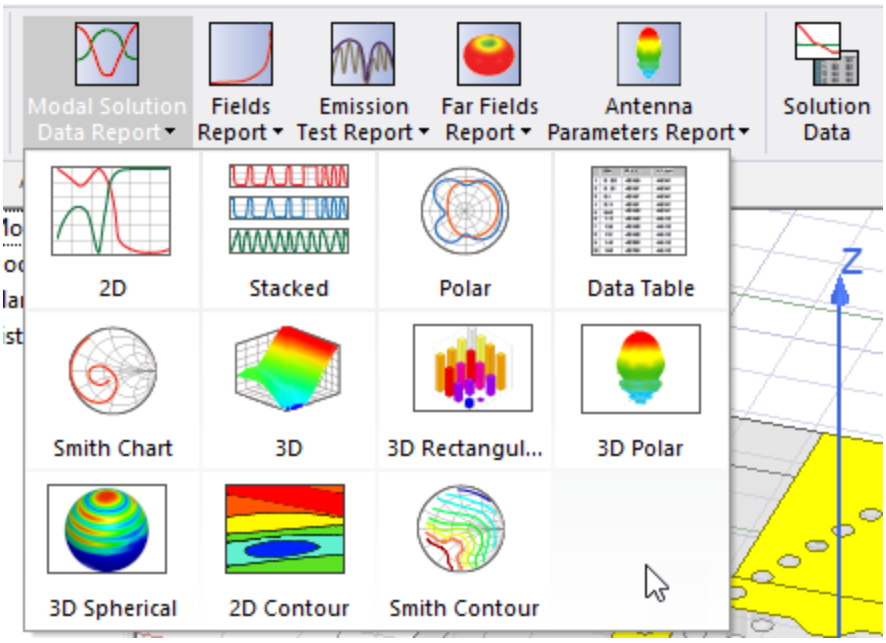
Selecting the Report Type

The **Report Types** available for creating a report depend on the simulation setup. The report category provides the default report name for the project tree and the text displayed in the report header. For example, S Parameters as the Category type causes the default report name to be S Parameter Plot *n*. Depending on the setup, you can make a selection from the following report types.

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.



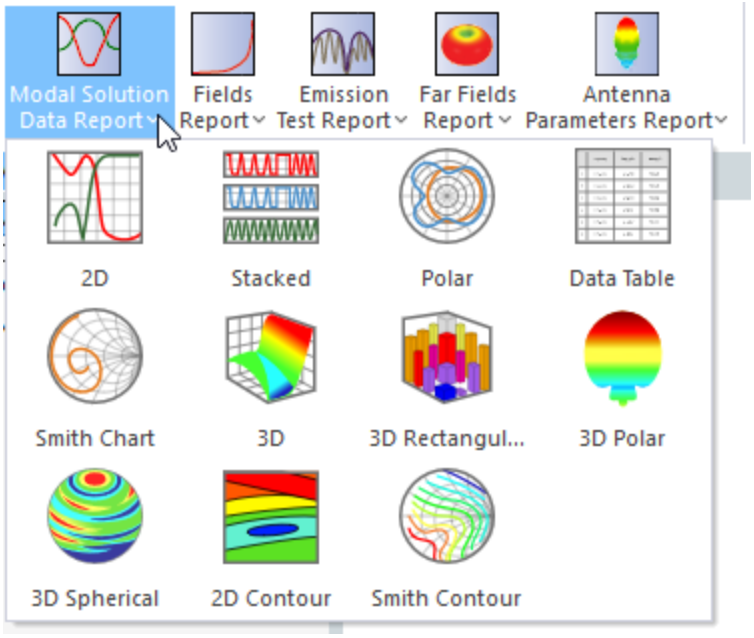
Report Type	Display Type
Create Standard Report	A Standard report can be depicted in any of the following display types: • Rectangular Plot • Data Table • Polar Plot • Smith Chart • Rectangular Stacked Plot • 3D Rectangular Plot • 3D Rectangular Bar Plot • 3D Polar Plot • 3D Spherical Plot • Rectangular 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:



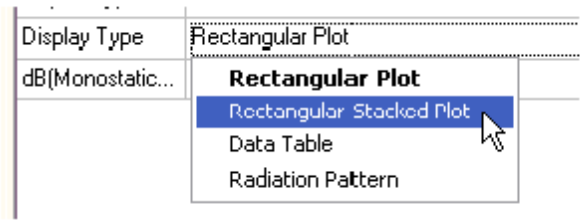
For the initial plot, you can also select from the following **Display Type** formats in the **Create*<type>* Report** submenu:

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 **HFSS 3D Layout** 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**.

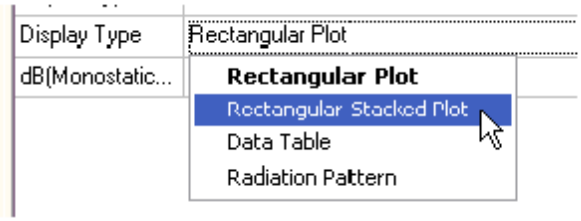
    This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog.

    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.

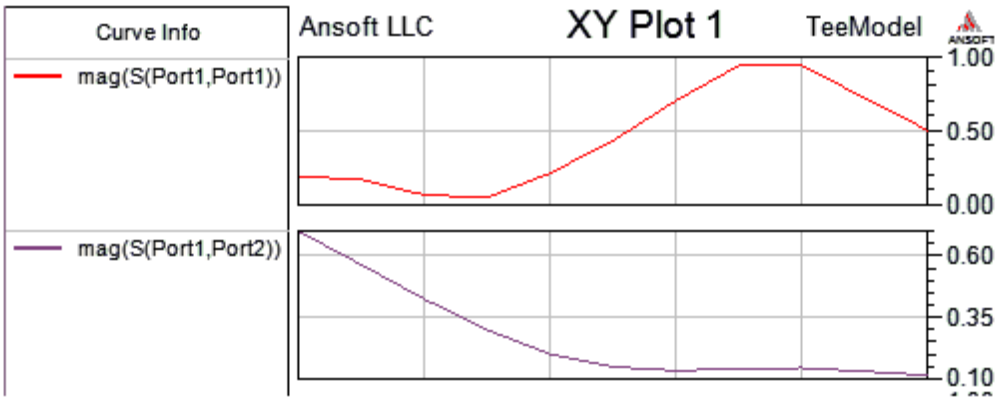
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 **HFSS 3D Layout** 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.**

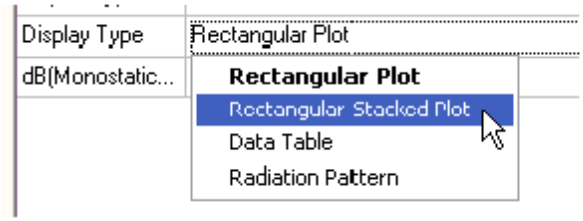
   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project 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.

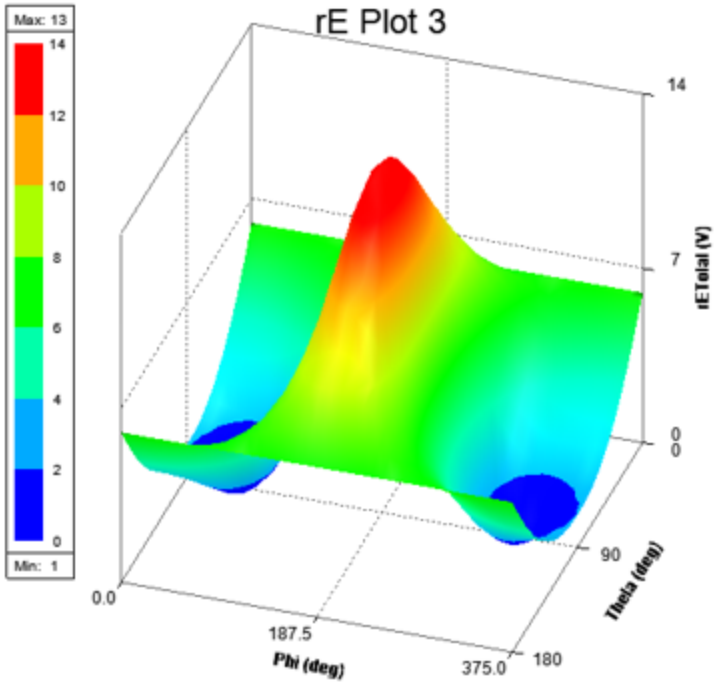
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 facing page.

Creating a 3D Rectangular Plot

1. On the **Results** menu (under the **HFSS 3D Layout** 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.

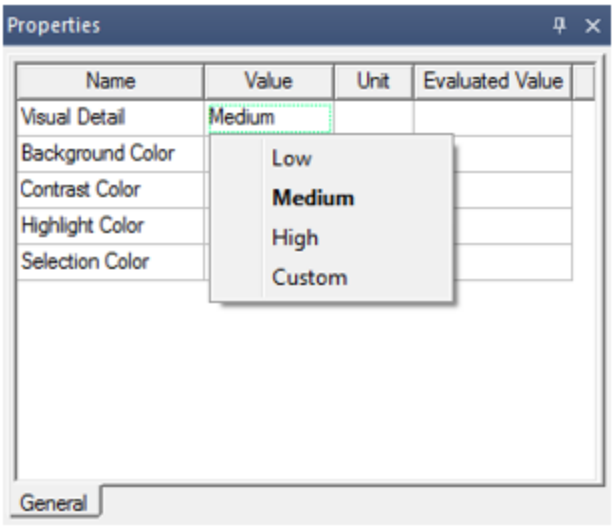
   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are 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.

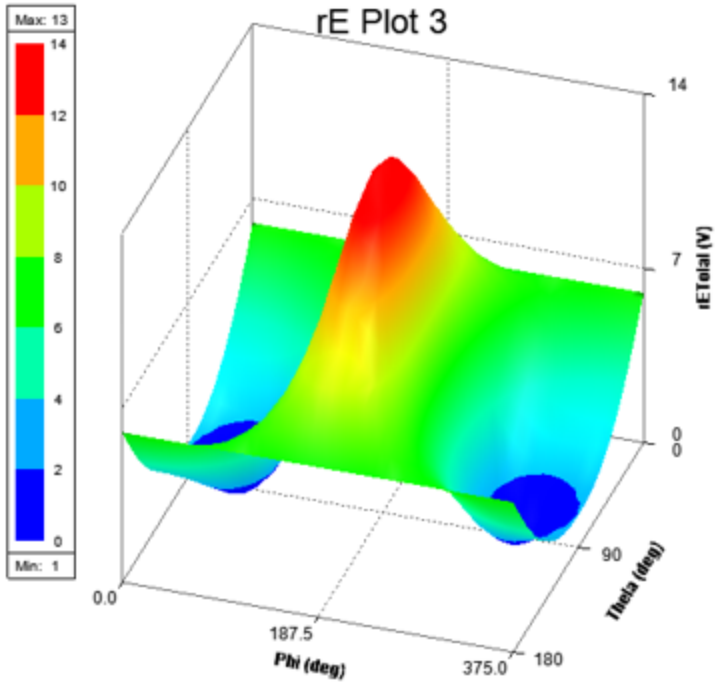


The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail 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

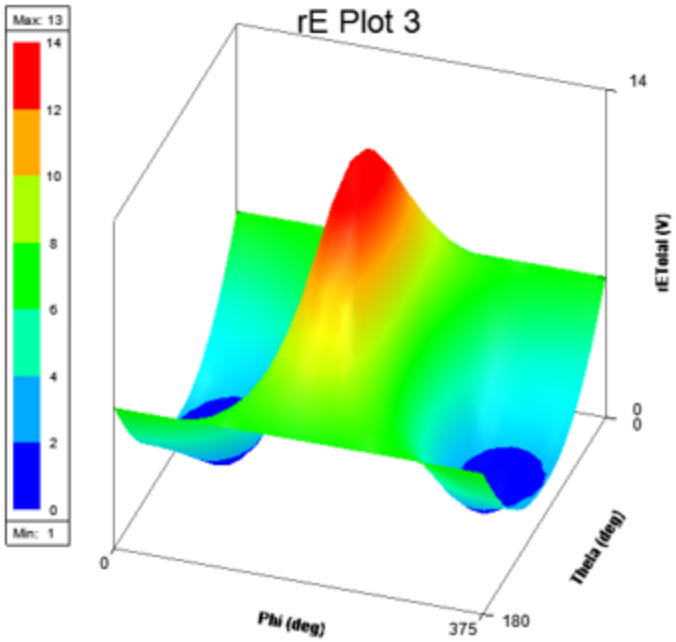
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

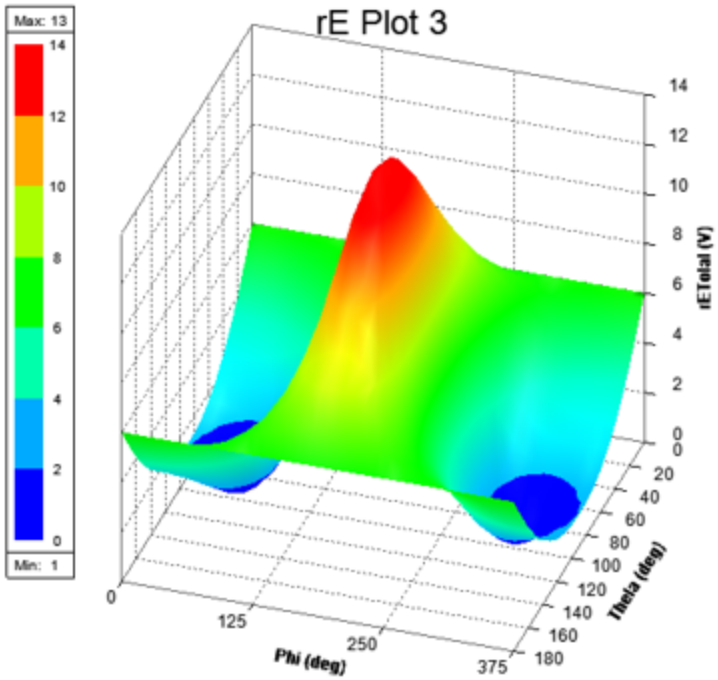
3D Rectangular Plot with Low Visual Detail

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

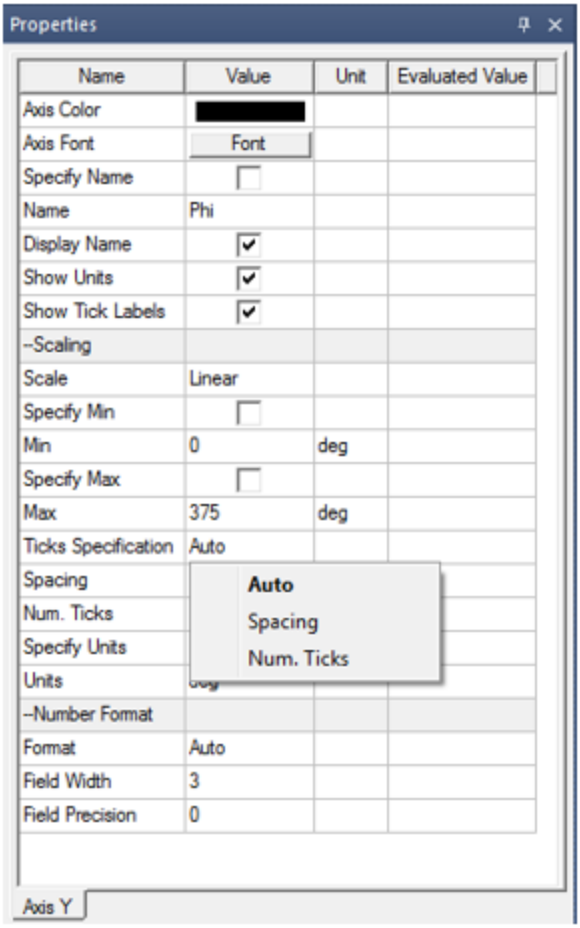


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 (**HFSS 3D Layout** 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.

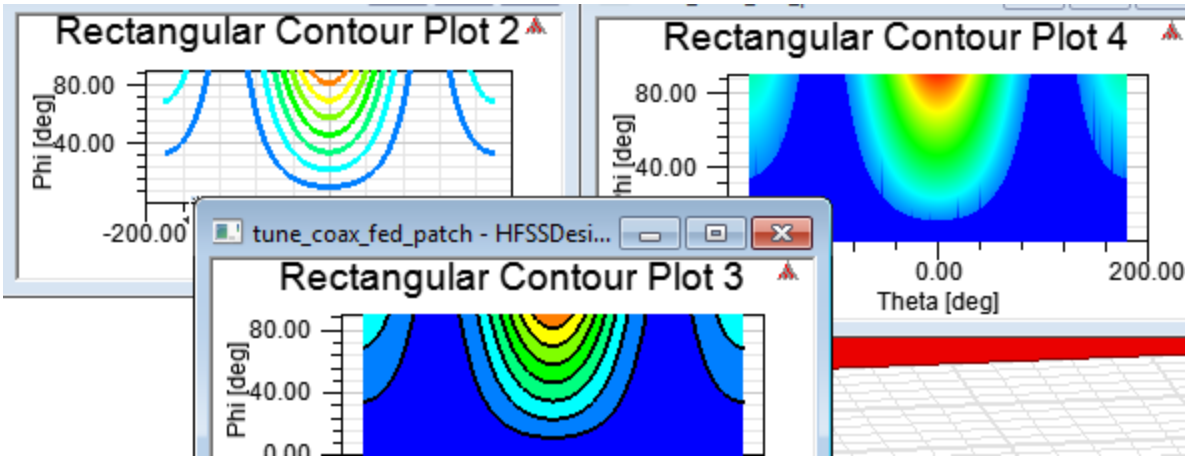
The Trace Properties for Cartesian plots include tabs for:

- 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

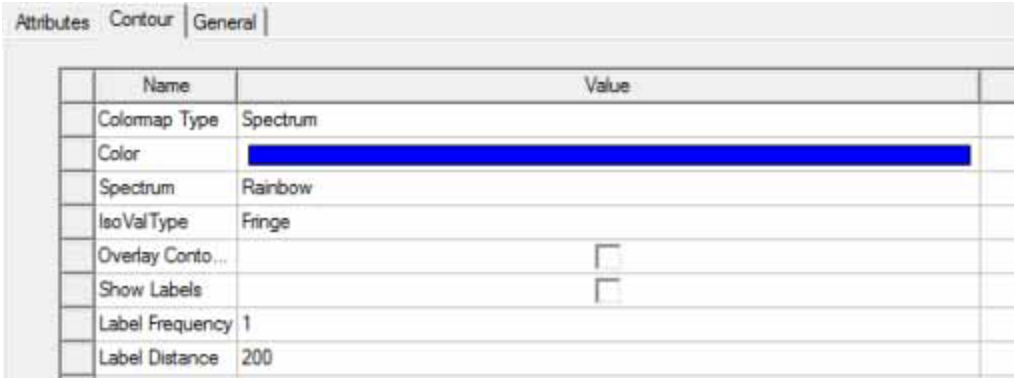
IsoValType can be Line, Fringe, or Tone.



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 Sine Space Plots

Sine Space plots provide an alternate way to view radiated field quantities. The transformation from theta, phi space to u, v sine space is defined as follows:

u = sin(theta)*cos(phi)

v = sin(theta)*sin(phi)

Note that this is similar to a polar coordinate system, where the radius is given by sin(theta) and the angle is given by phi. Another way to think about this: Given theta and phi that define a unit sphere, then these u,v coordinates are like viewing the unit sphere from above or below the XY plane. So, a sine space plot is a 2D plot that lies on a unit circle in the u,v space, that shows the values of radiated field quantities projected onto the upper or lower half of a full 3D sphere.

Sine Space Plots

Sine Space plots are implemented as a new Context in Reporter. They will leverage the existing radiated field setups and calculations.

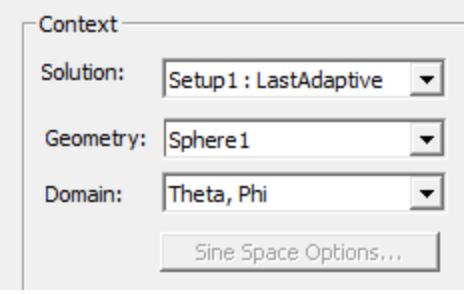
To create a Sine Space plot, after you have specified radiated field Setups.

1. Click **Results> Create Far Fields Report> Rectangular Contour Plot** or **Data Table**

   Results> Create Near Fields Report> Rectangular Contour Plot or **Data Table**.

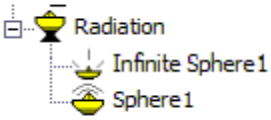
   The *Report* dialog box displays and shows choices for Domain in the Context field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



"Theta, Phi" is selected by default and gives the existing functionality.

Notice that the Context shown above already includes selection of a Geometry, which corresponds to a Radiation setup.

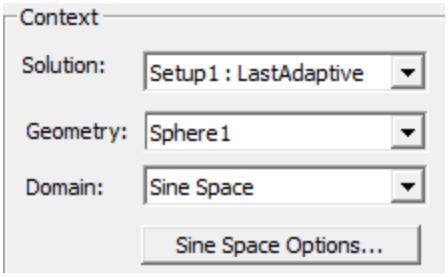


This radiation setup is used to calculate the fields for the Sine Space plot. Internally, this computes the values of the complex E vector based on the theta, phi sampling in the radiation setup. When data is requested for a sine space plot, it will correspond to some arbitrary theta, phi, and the fields will be interpolated to this location on the sphere. Note that this interpolation will occur on the complex E vector, and can be more detailed than a simple linear interpolation between sample points.

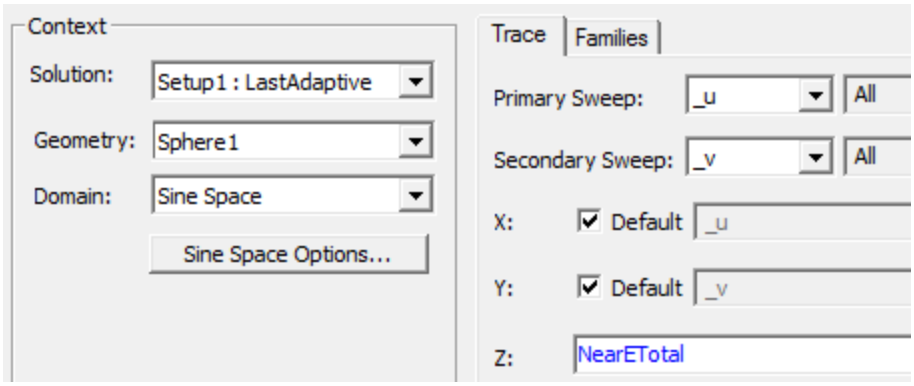
In order to support sine space plots, the radiation setup must have appropriate sampling in theta and phi. By default, a sphere is defined with phi from 0 to 360 (angle of rotation about Z), and theta from 0 to 180 (angle of rotation away from Z). These settings support a full unit circle representing the upper (theta from 0 to 90) or lower (theta from 90 to 180) half space. If the range of theta is reduced to 0 to 90, it would be possible to support upper half space only. Other ranges of theta and phi will likely create odd sine space plots, such as a partial circle, and these are not allowed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

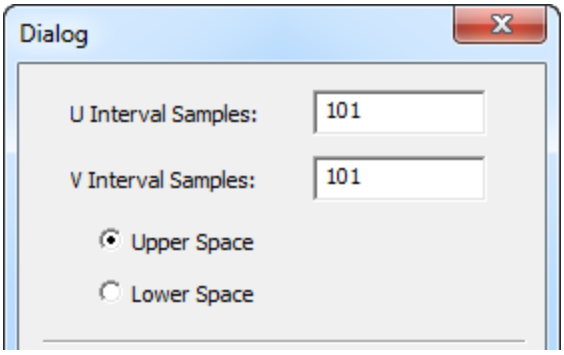
2. Select Sine Space as the Domain. This also enables the **Sine Space** options button.



Selecting "Sine Space" changes the Primary Sweep to U, and the Secondary Sweep to V.



The Sine Space Options provide additional settings, including the sampling interval for the u,v coordinates (default of .05 would provide 40 samples over each axis, -1 to 1) and selection of upper or lower half space (default to upper).

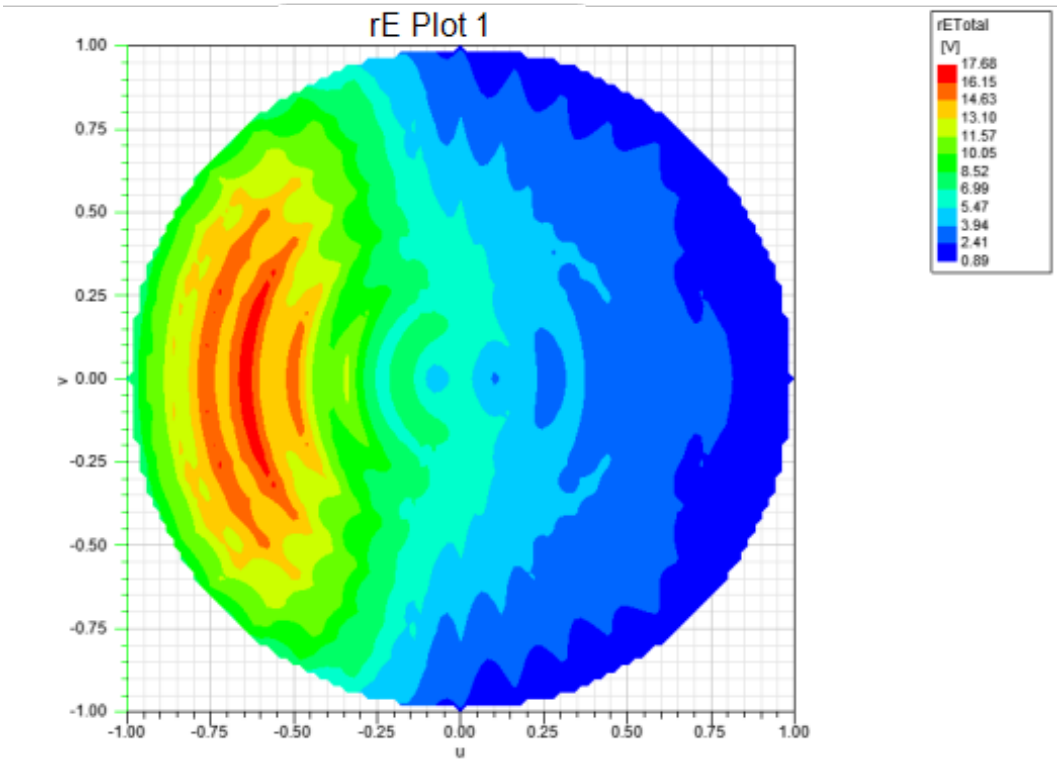


Once you have set up the domain, you define the Sine Space plot, selecting Category, Quantity and Function settings as appropriate. The Report Category you select provides the default report name, for example, S Parameter Plot *n* or Output Variables Plot *n*. Given

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

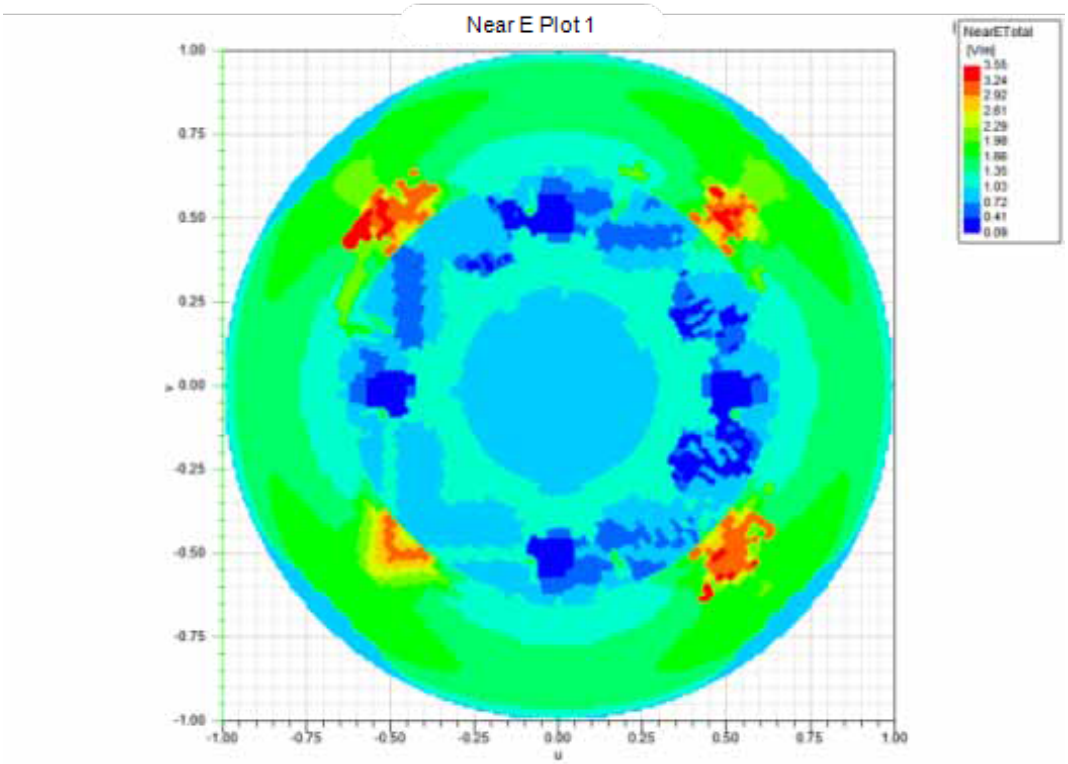
that the u,v sampling is set in the Sine Space Options, it should not be necessary to modify the sweeps in the Trace area of the dialog box.

Here is an example Sine Space plot of Far Field rETotal.



And here is an example Sine Space plot of Near field E Total.

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 **Contour** tab of the *Properties* dialog box to modify the appearance of the plot. See Creating Rectangular Contour Plots for discussion and examples.

Creating 2D Polar Plots

In HFSS 3D Layout, 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 (HFSS 3D Layout 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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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**.

Reviewing 2D Polar Plots

For a polar plot of S-parameters, HFSS displays in the lower-left corner the following derived information about the cursor's location:

MP	The magnitude and phase of the point.
RX	The normalized resistance (**R**) and reactance (**X**).
GB	An alternate view of the normalized resistance and reactance in the form of

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$R + jX = \frac{1}{G + jB}$$

where

- ***G*** = conductance
- ***B*** = susceptance

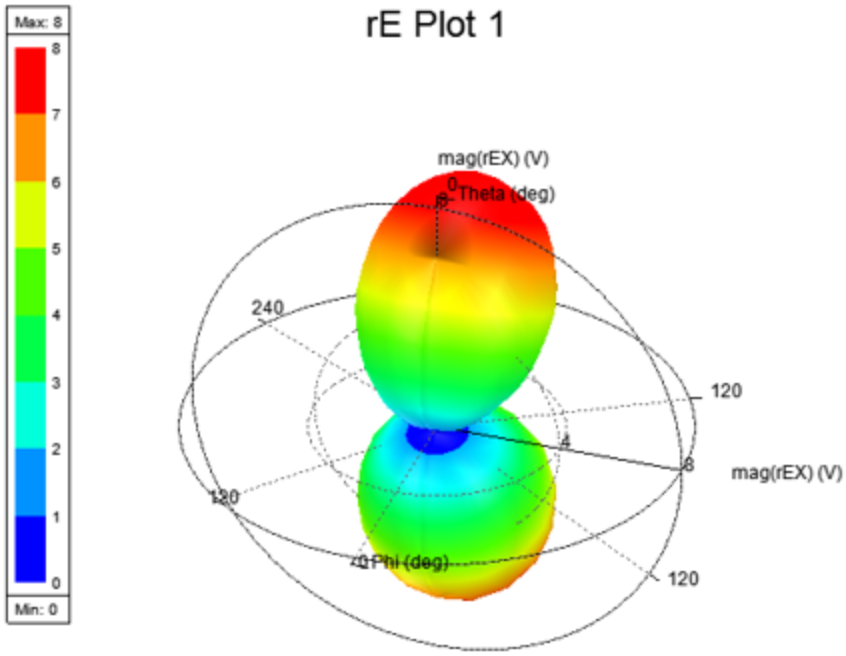
Q The quality factor.

VSWR The voltage standing wave ratio, calculated from the equation $\frac{1 + |S_{ij}|}{1 - |S_{ij}|}$.

A scale below the plot displays the scale of points along the R-axis.

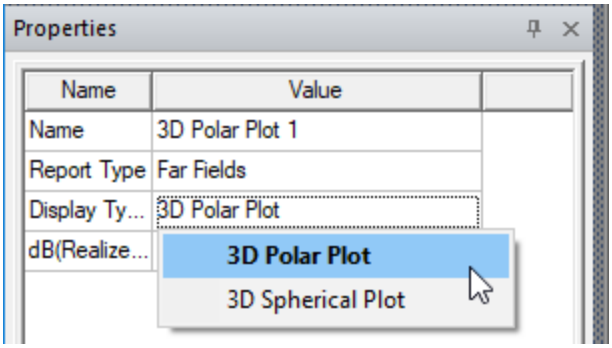
Creating 3D Polar Plots

A 3D polar plot is a 3D circular chart divided by the spherical coordinates R, theta, and phi, where R is the radius, or distance from the origin, theta is the angle from the x-axis, and phi is the angle from the origin in the z direction.

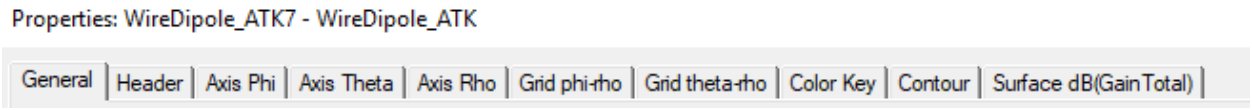


To convert a 3D Polar plot to a 3D Spherical Plot, select the plot in the Project tree, and select from the Display type menu in the Properties window:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Double-clicking anywhere in the plot brings up the display **Properties** dialog box. The properties are grouped appropriately under various tabs, which correspond to plot entities.



- General: For general plot properties such as Visual Detail level and background color
- Header: Properties related to plot Header/Title
- Axis Phi: Properties related to the circular axis which is in XY plane
- Axis Theta: Properties related to the circular axis which is in YZ plane
- Axis Rho: Properties related to the radial axis
- Grid phi-rho-theta(0): Properties related to phi-rho grid at theta = 0 (XY plane)
- Grid theta-rho-phi(90): Properties related to theta-rho grid at phi = 90 (YZ plane)
- Color Key: Properties related to the color key, 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

You can edit the properties on these tabs to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Polar Plot" on page 14-117 below.

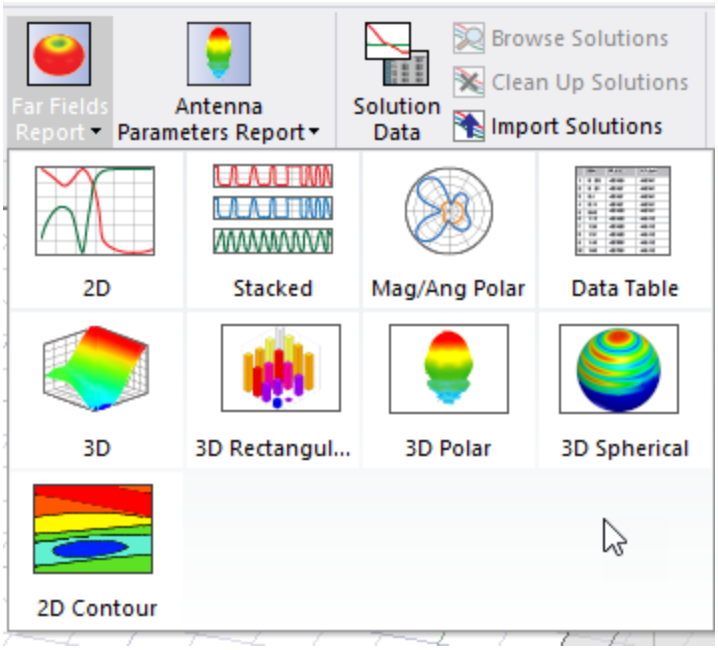
Creating 3D Polar Plots

Following is the general procedure for drawing a 3D polar plot of results:

1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create <*type*> Report**, and select **3D Polar Plot** from the report type menu.

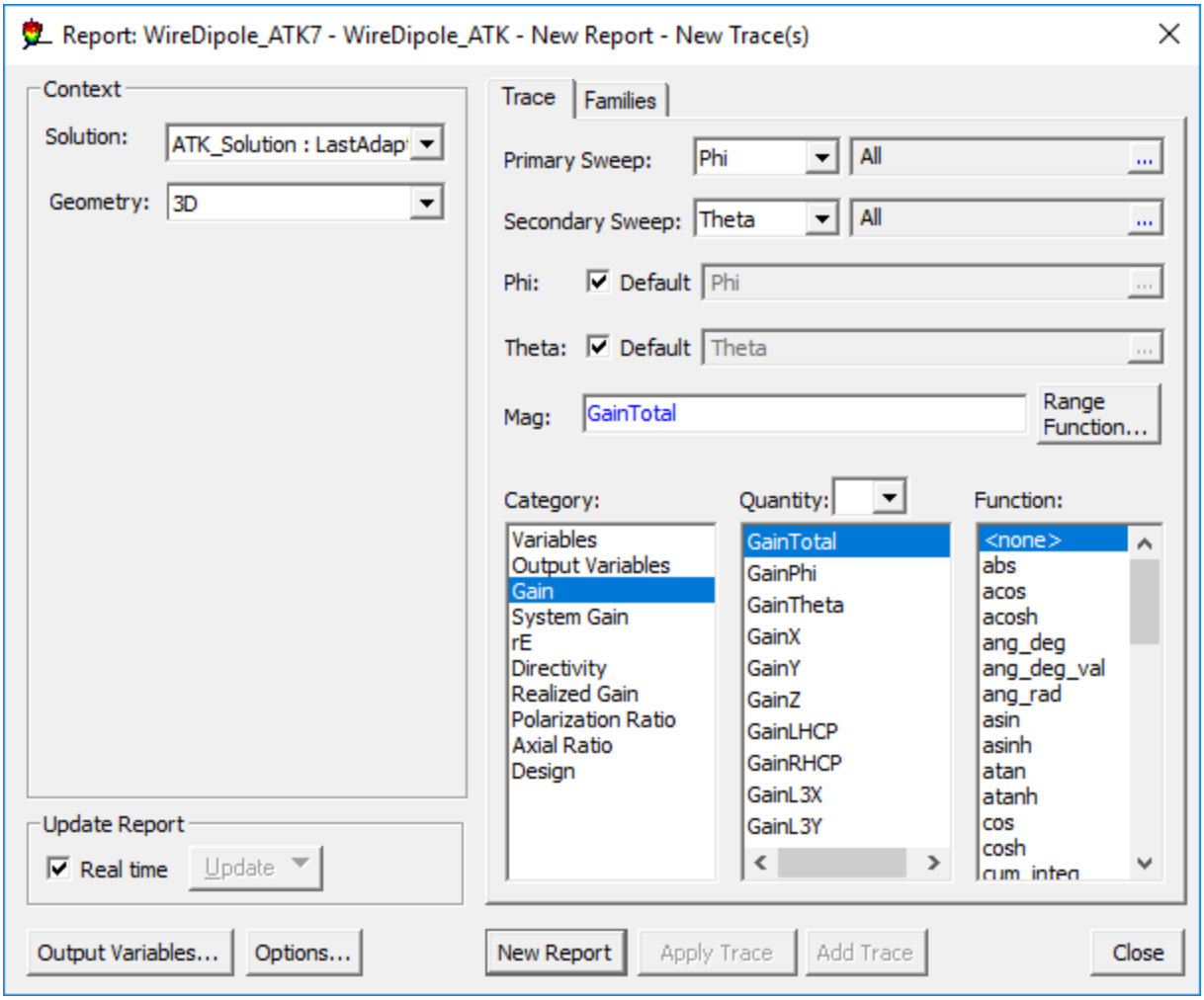
   You can also select 3D Polar Plots from the **Results** tab of the ribbon if such plots are appropriate for the current design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **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. 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. In the **Trace** tab **Mag** area, specify the information to plot along the R-axis, or the axis measuring magnitude:
    a. From the **Category** drop-down menu, select the type of information to plot. The selected category provides the default plot name.
    b. In the **Quantity** list, click the values to plot. Ctrl+click to make multiple selections.
    c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

4. On the **Trace** tab **Theta (Secondary Sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values to plot along the theta-axis.
5. On the **Trace** tab **Phi(Primary Sweep)** line, select the sweep variable from the drop-down menu, and specify all values or select values to plot along the phi-axis.
6. Click **New Report.**

   This button 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 R-, phi-, and theta-axes on a 3D polar graph. The plot is listed under **Results** in the project tree. 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

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Interacting with 3D Polar Plots

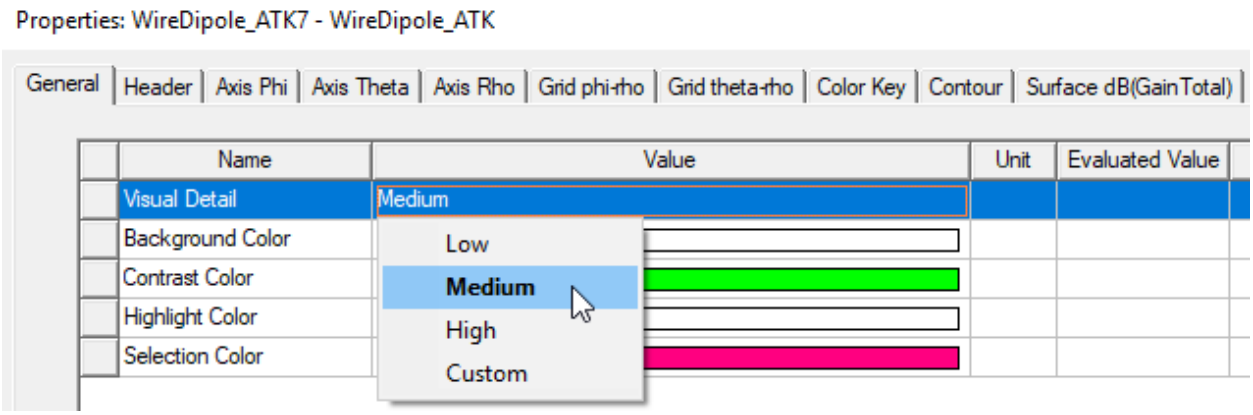
You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Controlling Visual Detail in a 3D Polar 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. Clicking on a plot entity selects it, highlighting the selected entity in bold.

The Visual Detail property on the **General** tab also provides control suited to different screen and plot sizes.

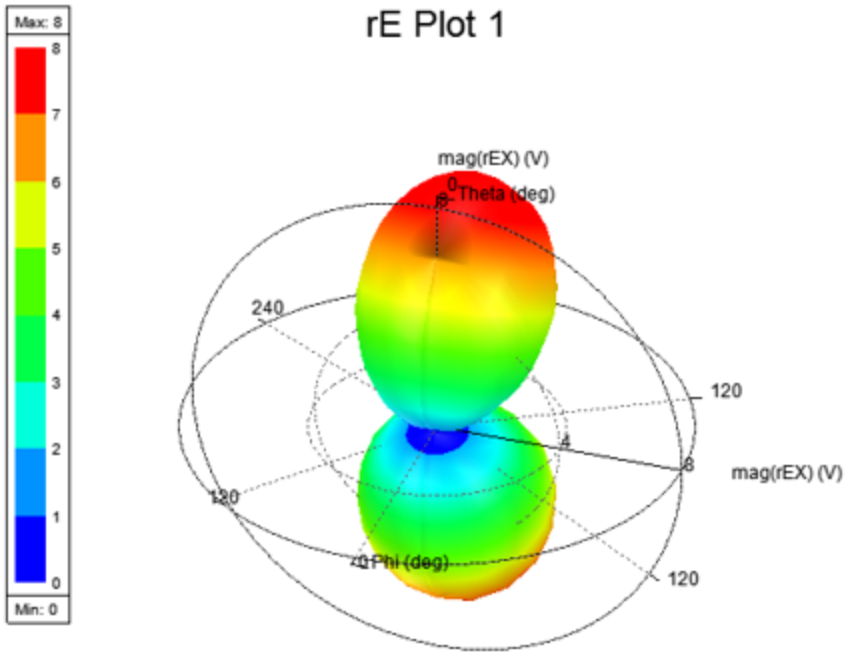
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

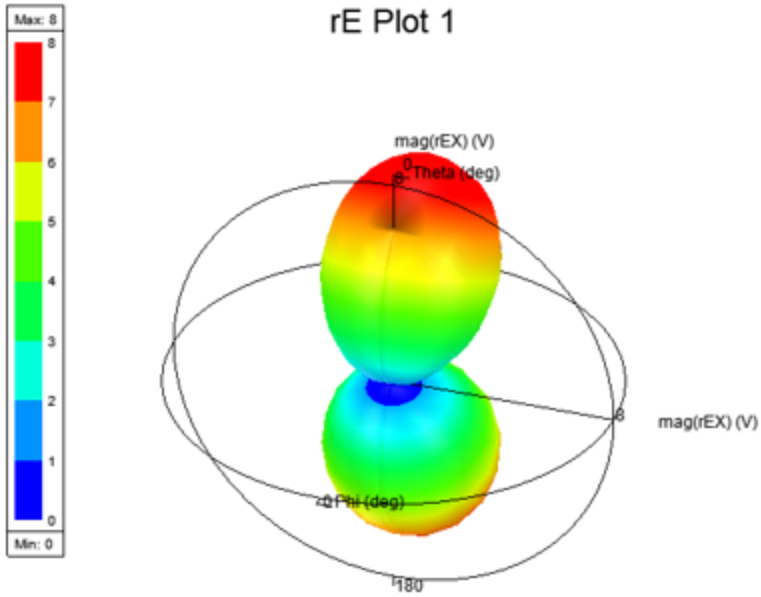
3D Polar Plot with Medium Visual Detail



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 Polar Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under the Medium Visual Detail level, 3D Polar Plot has 3 ticks per axis (phi, theta and rho axis) which show min, max and middle value. This setting also shows axes labels.

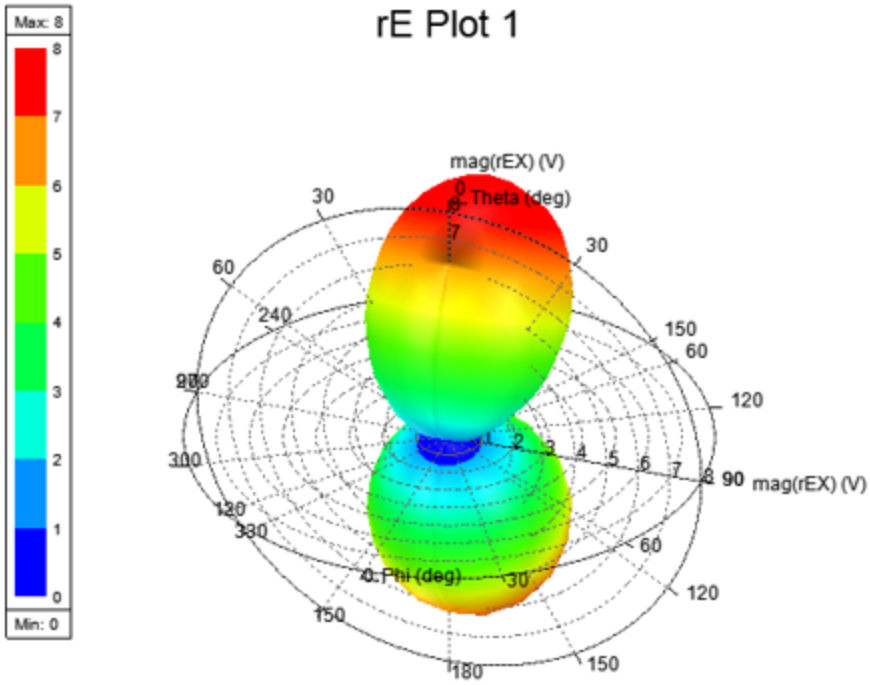
3D Polar Plot with Low Visual Detail



With Visual Detail set to Low, 3D Polar Plot does not show polar grids or grid lines. It only shows axes with 2 ticks corresponding to min and max values. This setting also renders axis labels.

3D Polar 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.

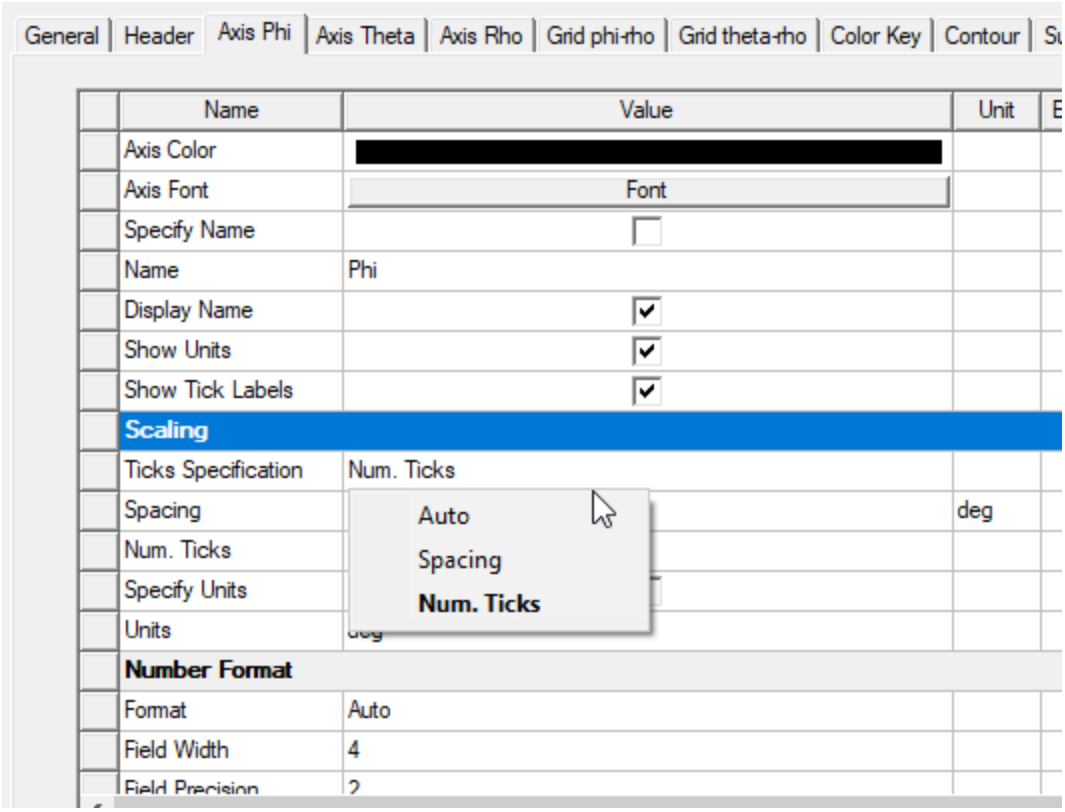


With Visual Detail set to High, a 3D Polar Plot shows all polar axes and grids together with all nice ticks and axes labels. This is ideal for large plot sizes.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, 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.



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:

--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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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** Unchecked, Ticks Specification is set to Auto.
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** Checked, Ticks Specification is set to Spacing.

Grid Properties for Phi-Rho and Theta-Rho

You can control a range of display properties for the pho-rho and theta-rho grids, including line styles and colors for major and minor grids.

Overlay 3D Polar Plot on Model Window

For a 3D polar plot to be eligible for overlay, it must have its primary and secondary sweep from variables Phi and Theta or IWavePhi and IWave Theta in that order. If the plot is unsuitable, the **Overlay** commands are disabled.

Once you create a suitable plot, you can overlay the 3D polar plot on the model window.

Creating Smith Charts

A Smith chart is a 2D polar plot of S-parameters upon which a normalized impedance grid has been superimposed. Following is the general procedure for creating a Smith chart of results:

1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create <*type*> Report**, and select **Smith Chart** from the report type menu.

   The **Report** dialog appears.

2. 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. The category selected provides the default plot name.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

3. 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 values you specified on a polar plot. In addition, each circle on the plot is labeled with values of *R*, measuring normalized resistance, and each line is labeled with values of *X*, measuring normalized reactance. 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.

4. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**.

Creating Smith Contour Charts

A Smith contour chart is a polar plot of S-parameters upon which a normalized impedance grid has been superimposed. Following is the general procedure for creating a Smith chart of results:

1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create <*type*> Report**, and select **Smith Chart** from the report type menu.

   The **Report** dialog appears.

2. In the **Trace** tab **Mag** area, specify the information to plot:
   a. On the **Category** drop-down menu, click the type of information to plot. The Category selection provides the default plot name.
   b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

3. On the **Trace** tab **(Secondary Sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values to plot along the theta-axis:

   To select an Secondary sweep component that is different from the default, 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.

   a. If sweeps are available, you can select **Browse** [...] to display a dialog that lets you select particular sweep or sweeps, or all sweeps.

   b. 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.

4. On the **Trace** tab **(Primary Sweep)** line, select the sweep variable from the drop-down menu, and specify all values or select values to plot along the phi-axis:

   To select an X component that is different from the default, 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.

   a. If sweeps are available, you can select **Browse** [...] to display a dialog that lets you select particular sweep or sweeps, or all sweeps.

   b. 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. Click **New Report.**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 values you specified on a polar plot. In addition, each circle on the plot is labeled with values of *R*, measuring normalized resistance, and each line is labeled with values of *X*, measuring normalized reactance. 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.

6. 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 **HFSS 3D Layout** > **Results** > **Create *<type>* Report** or right-click **Results** in the Project Manager and click **Create *<type>* Report**.
2. In the **display type** menu, click **Data Table**.

   The *Report* dialog box appears.

3. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed depend on the Solution type and the report ***<type>*** selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.
   c. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
4. Under the **Trace** tab **Y** component section, select the quantity you are interested in and its associated function:
   a. On the **Category** drop-down menu, click the type of information to plot. The category selected provides the default name for the plot.
   b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
   c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.
   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.

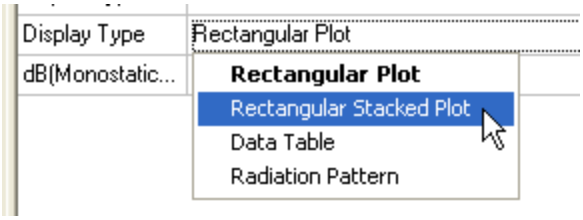
5. On the **Trace** tab **X (Primary sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values.
6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The Y quantity will be listed at each variable value or additional quantity value you specified. The data table is listed under **Results** in the project tree. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

If you choose to print a data table:

- Selecting print "All" prints the whole table for current data page (if there is more than one data page).
- Selecting print "Pages" prints user-specified pages.

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 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.

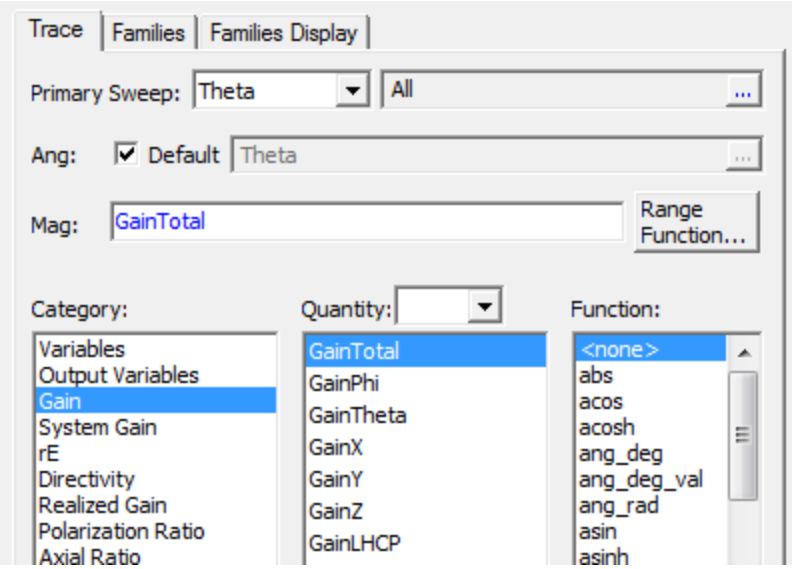
Creating Radiation Patterns

A radiation pattern is a 2D polar plot displaying the intensity of near- or far-field radiation patterns. It is 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 radiation pattern of results:

1. Click **HFSS>Results>Create Far Fields Report>Radiation Pattern**, or right-click the **Results** icon in the **Project tree** and click **Create Far Fields Report>Radiation Pattern**.

The **Report** dialog box appears, and a Radiation Pattern Plots icon appears under **Results** in the Project tree.

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.
4. In the **Trace** tab area, specify the Primary Sweep and Ang information to plot along the R-axis, or the axis measuring magnitude. The Primary Sweep field has a drop-down menu of current choices for Theta, Phi, and defined variables. Click the ellipsis button if you want to select from available values.
5. In the **Ang**, if you want to change the default, uncheck to enable the field and ellipsis button.



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 selected category also provides the default plot name.
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 **Mag** 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.

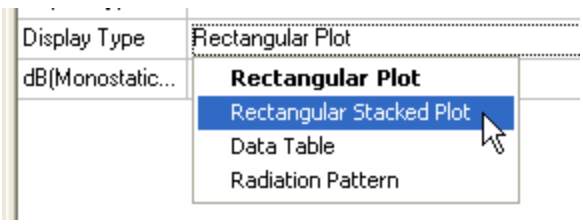
5. The **Families** tab is helpful if you want to use the plot as an overlay and you need to restrict the Phi (default) or Theta values to make the plot appropriate for overlay. See Overlaying 2D Radiation Field Plots on Models.
6. Click **New Report.**

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, System Gain Plot *n* or Directivity 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 a 2D polar plot. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Plotting in the Time Domain

The idea behind Time-Domain Reflectometry (TDR) is to excite a structure with a step function, and inspect the reflections as a function of time. Before you can examine the time domain, you must perform an Interpolating sweep for a driven solution (Modal or Terminal or Transient). You can then select **Time** from the **Domain** list in the **Report** dialog box. You also need to specify the input signal, whether step or impulse.

With **Time** selected as the domain, you can select from several Categories and associated Quantities to plot, for example mag(S_{11}). When you plot in the Time domain, every frequency domain quantity is first converted to the time-domain before the formula is evaluated. For example, if you type in

$S_{11} / (1 - S_{11})$

and plot it in the time domain the reporter will plot

IFFT(S_{11} * input)/(IFFT(input)-IFFT(S_{11} * input))

It will NOT plot

IFFT(S_{11}/ (1 - S_{11}) * input)

The two expressions are not equivalent.

If you select Time Domain Impedance as the Category, you can select the TDRZ quantity. This is defined as

TDRZ(t)=Zref*(IFFT(input)+IFFT(S_{11} * input))/(IFFT(input)+IFFT(S_{11} * input))

where "input" denotes the Fourier transform of the input signal (step or impulse) and "IFFT(.)" denotes the inverse FFT.

This equation is the instantaneous ratio of the time-domain voltage v(t) to the time-domain current i(t). That is because voltage and current are defined (in the frequency domain) in terms of the incident and reflected waves a and b, respectively, as

$V = sqrt(Zo) * (a + b) = sqrt(Zo) * (1 + S_{ii}) * a$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$I = 1/\text{sqrt}(Zo) * (a - b) = 1/\text{sqrt}(Zo) * (1 - S_{ii}) * a$

This lets the incident wave be the input step signal, and so when we take the inverse FFT of V and I, we get v(t) and i(t) in the time domain. Taking their ratio as a function of time then yields TDRZ(t). By default, Zo is equal to 50 Ohm. For frequency dependent Z_0, such as from an HFSS waveport, we recommend renormalizing to a constant Z_0, such as Z_0 = 50 ohms. This mimics the measurements techniques from which TDR was developed.

For HFSS, HFSS 3D or Layout Linear Network Analysis on Circuit, to create a plot in the **Time Domain**:

1. For a design with an existing sweep setup, follow steps 1 - 4 for creating a report for design.
2. In the **Report** dialog box, in the **Domain** list, click **Time**.

   This enables the **TDR Options** button and for terminal solution data reports includes the Terminal TDR Impedance in the Category list.

3. Click the **TDR Options button**.

   The **TDR Options** dialog box appears.

4. Select the input signal type, **Step** or **Impulse**.

   A **Step** describes a sustained change in the signal, whereas the **Impulse** is a brief excitation. **Impulse** is a very narrow rectangular pulse, with zero rise and fall time, width of 1 time step, and height of 1/(time step).

   Selecting **Step** enables the **Rise Time** field, and **Impulse** disables it.

5. If you selected **Step**, enter the rise time of the pulse in the **Rise Time** text box.

   The rise time should be appropriate for the frequency context.

   With a band width from DC to f_{max}, the best time resolution that can be achieved is $1/(2f_{max})$.

   A rise time of $1/(2f_{max})$ is the shortest rise time that can be resolved. However, a rise time of 0 s gives equally valuable information, so 0 is the default in this panel. See the example plot.

6. Enter the total time on the plot in the **Maximum Plot Time** text box.

   The default maximum plot time in the **TDR Options** dialog is related to the delta frequency df in the frequency sweep: it is 1/2df, since that is the extent of time for which the IFFT gives information. This is often very long relative to the time delay that corresponds to the length of your device under test, so you may want to reduce this value. Alternatively, you can adjust the time axis of your TDR plot after it has been created.

7. Set the number of time points to plot in the **Delta Time** text box. By default, this is set to the number of points in the frequency sweep.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The delta time is based on the bandwidth of the sweep: with a frequency sweep from DC to f_{max}, the smallest time resolution you can obtain is given by $1/(2f_{max})$. The IFFT algorithm provides data points as a spacing of $1/(2f_{max})$, but you can smoothly interpolate between points by setting a finer resolution, e.g., $1/(10f_{max})$, at the expense of extra computation time.

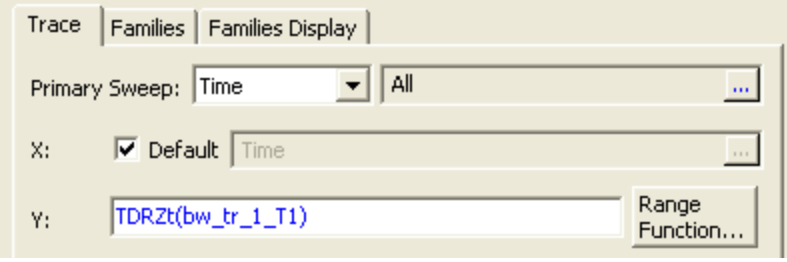
8. Optionally, under **TDR Window**, modify the window type and width.
9. You can use the **Save as Default** to set the current values as a default, and the **Use Defaults** button to use previously saved options. Note that when you select a trace, the initial displayed values are those of the selected trace.
10. Click **OK**.

Optionally, to plot Terminal TDR impedance (that is, rather than calculate the S-parameter for waveport1 versus frequency, instead calculate the delay versus time at a particular impedance), do the following:

a. In the **Category** list, click **Terminal TDR Impedance**.
b. In the **Quantity** list, click a quantity to plot.

The default impedance (Zo) for the TDRZ quantity is 50 Ohms, unless you specified differently when you Set Renormalizing Impedance for Terminals when you created the terminals in the model. If you need a different impedance value, you can either edit the value in the **Report** dialog (as shown below), or you can create an Output Variable representing Zo × $(1+S_{ii})/(1-S_{ii})$ with the Zo of your choice. To edit the Zo value in the **Report** dialog:

1. For the Category, select Terminal TDR Impedance, and the Port and Function of interest.



2. Edit the value by placing the cursor in the Value field.

In this example, the value for Zo is changed from the default to 75 Ohms by typing ',Zo=75ohm' in the Y-column field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

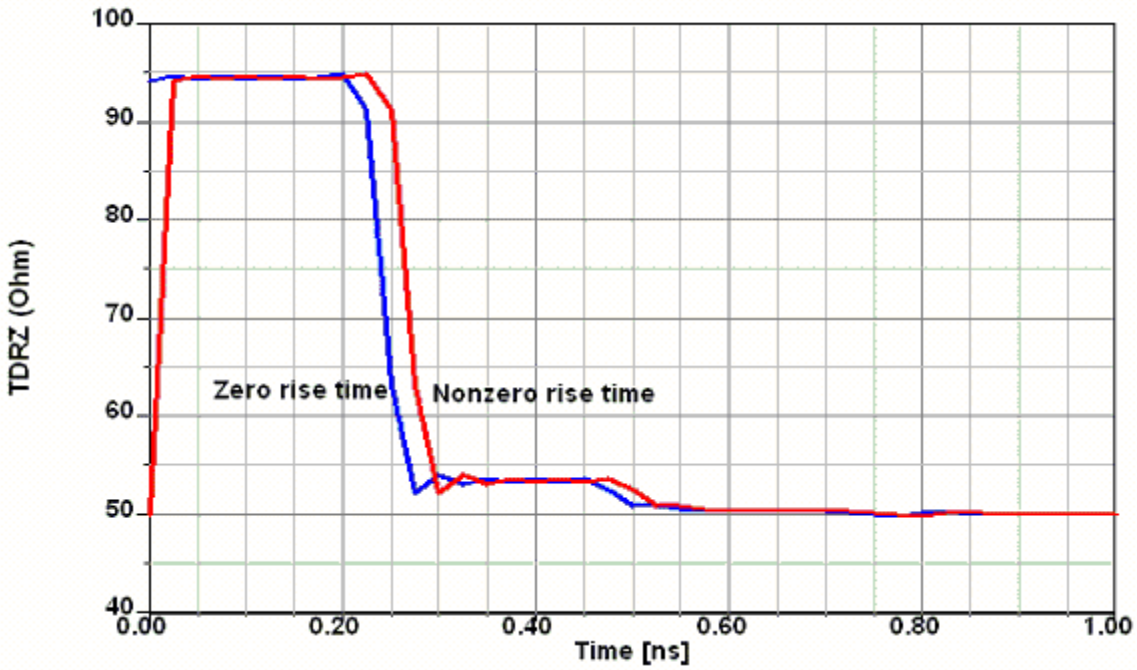
c. In the **Function** list, click the mathematical function of the quantity to plot.

3. Click **Done**.

The report appears in the view window. It will be listed in the project tree.

If S_{11} = 0 at DC, the time-domain step response will settle to zero and the TDRZ step response settles to Zref. If S_{11} is nonzero at DC, the time-domain step response will settle to a nonzero value and TDRZ will settle to a value different from Zref. The time-domain impulse response will always settle to zero, since it can be seen as the derivative of the step response. The TDRZ impulse response will always settle to Zref.

The plot below shows the difference between a short nonzero rise time and zero rise time for a transmission line segment of 94 Ohm. Note that the trace with zero rise time starts at the correct line impedance while the other starts at the renormalizing impedance. Other than that, one trace is a shifted version of the other. The reason the plot with finite rise time starts at 50 ohms is that the time-domain voltage and current are still at their steady state values, so v = Zref * i. As the pulse arrives, the TDRZ response changes from the steady-state behavior because there's a reflection from the transmission line back to the exciting source, which has a different renormalizing impedance from the characteristic impedance of the transmission line.



Some things to keep in mind with TDR:

$$Spatial resolution \Delta x = c/(2B) \quad (1)$$

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 speed of light in the medium and B is the bandwidth of the signal. Since TDR is usually based on a frequency band that starts at DC, the spatial resolution becomes

$$\Delta x = c/(2F_{max}) \quad (2)$$

where F_{max} is the highest frequency in the frequency sweep. For example, if F_{max} = 15 GHz and the medium has ε_r=4, the spatial resolution will be (1.5E8 m/s)/(3E10 /s) = 5 mm.

A spatial resolution of $c/(2F_{max})$ corresponds to a resolution in time

$$\Delta t = 1/(2F_{max}) \quad (3)$$

Let N be the number of points in the IFFT. N equals the number of time samples, and it also equals twice the number of frequency samples. The density of frequency samples in the frequency sweep influences the total time T as follows:

$$2F_{max}/(\Delta f) = N(\text{number of points in IFFT}) = T/\Delta t \quad (4)$$

So increasing the density of the frequency samples leads to an increase in total time T. In practical case, this often leads to a long tail in the TDR plot with little useful information. Therefore, the TDR Options interface lets you set the maximum plot time to a smaller value.

The TDR Options interface also lets you choose a smaller Δt than given by equation (3) above. When you choose a smaller Δt, you increase F_{max} by "zero padding" (adding zero values for S_{11} beyond the calculated frequency sweep). Whether this is justified depends on your judgment. It leads in practice to a smoother TDR signal.

HFSS also lets you set the rise time of your input signal. The rise time should be at least $1/(2F_{max})$. Even this rise time is a bit short for comfort, as it equals the duration of only one time sample. An input signal with a longer rise time has a smaller high-frequency content and will lead to reduced "ringing" in the TDR response.

A Hamming or Hann filter will also reduce the high-frequency content and tends to lead to a smoother TDR response. With these filters, one can select a width. A width of 100% is often a good choice.

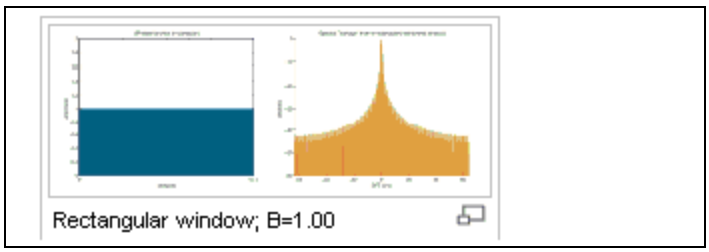
TDR Windowing Functions

Windowing functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ability to resolve signals of different strengths and frequencies. The window type list includes the following:

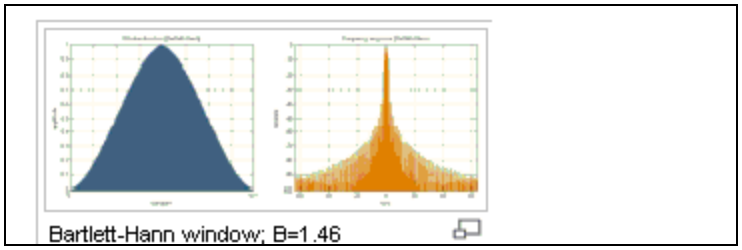
Window Function	Preferred Use
Rectangular	A low dynamic range function offering good resolution for signals of comparable strength. Poor when signals have very different amplitudes. $w(n)=1$.



Bartlett — A high dynamic range function, with lower resolution, designed for wide band applications.

$$w(n) = a_0 - a_1 \left| \frac{n}{N-1} - \frac{1}{2} \right| - a_2 \cos\left(\frac{2\pi n}{N-1}\right)$$

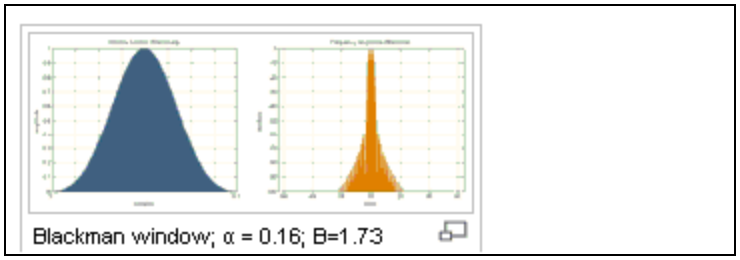
where a_0=0.62; a_1=0.48; a_2=0.38



Blackman — A high dynamic range function, with lower resolution, designed for wide band applications.

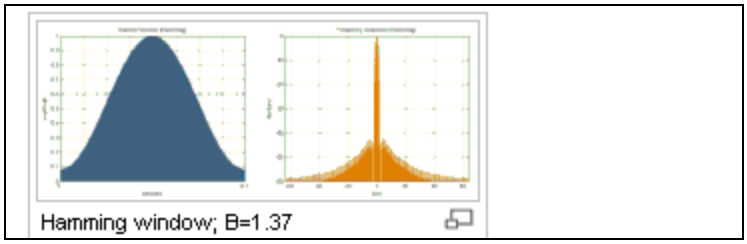
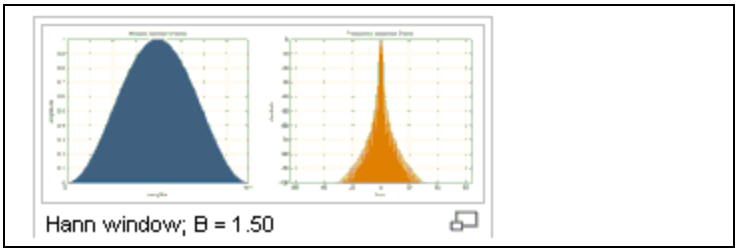
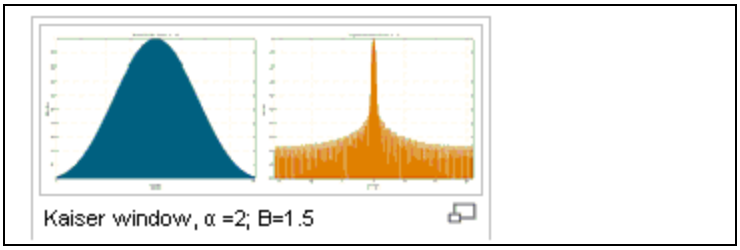
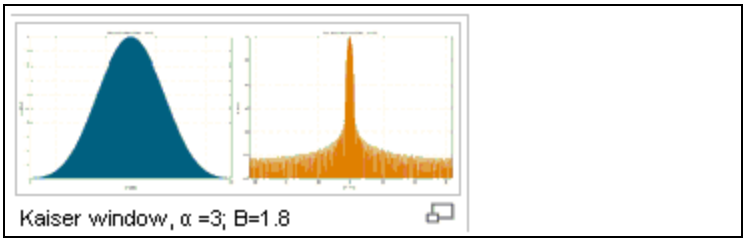
$$w(n) = a_0 - a_1 \cos\left(\frac{2\pi n}{N-1}\right) + a_2 \cos\left(\frac{4\pi n}{N-1}\right)$$

where $a_0 = (1-\alpha)/2$; $\alpha_1 = 1/2$; $\alpha_2 = \alpha/2$



Hamming — A moderate dynamic range function, designed for narrow band applications. It minimizes the maximum sidelobe.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use
	$w(n) = 0.54 - 0.46\cos\left(\frac{2\pi n}{N-1}\right)$ 
Hanning (default)	A moderate dynamic range function, designed for narrow band applications. $w(n) = 0.5\left(1 - \cos\frac{2\pi n}{N-1}\right)$ 
Kaiser	Selecting the Kaiser plot also enables a field to specify an associated Kaiser parameter. The larger the Kaiser parameter, the wider the window. The parameter controls the trade off between width of the central lobe and the area of the side lobes.  
Welch	This approach applies a parabola-shaped window to the frequency domain data. It is based on the Bartlett method but splits the signal into overlapping segments, which are then windowed. The intent is to balance the influence of data in the center of the function.

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 **Save as Default** to set the current values as a default.

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:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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):

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.
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:

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.)

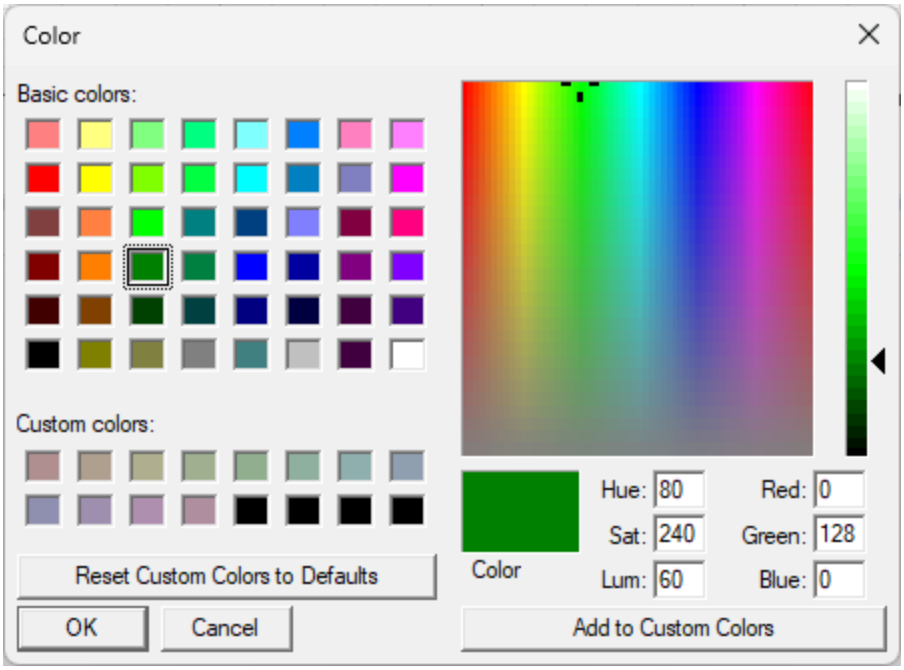
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

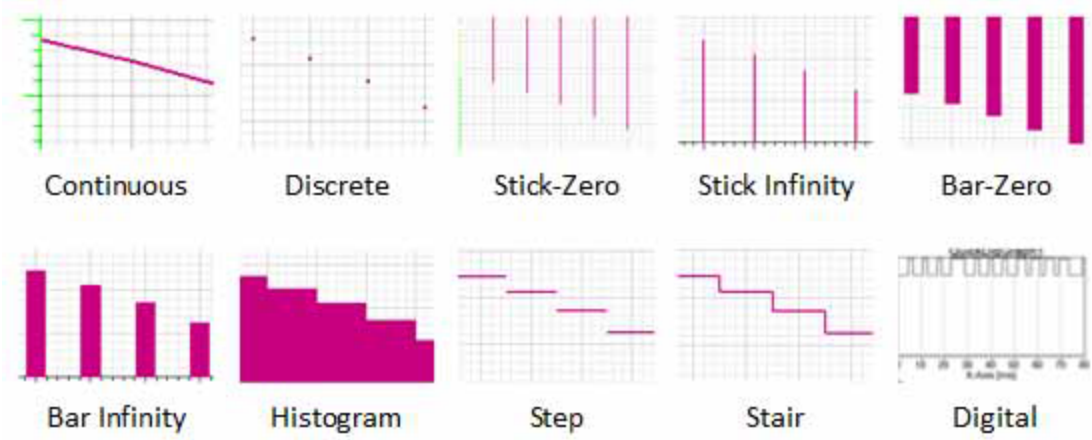
  To edit a trace name, see Editing Trace Properties.

- 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.



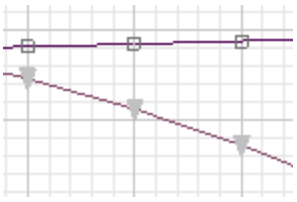
- Line style – a drop-down menu lets you select Solid, Dot, Dash, or Dot-dash.
- Line width – a text field lets you edit the numeric value.
- Trace type – the drop-down menu contains entries for Continuous, Discrete, Bar-Zero, Bar Infinity, Stick Zero, Stick Infinity, Histogram, Step, and Stair.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

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.

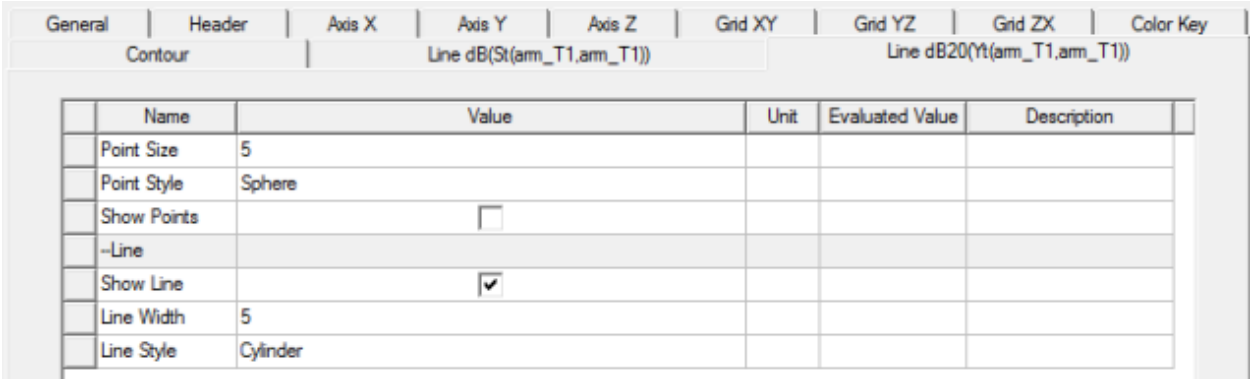
> **Note:**
>
> So that curves with single points always appear, Box is the default symbol.

3. Edit the properties, if needed. Click OK to apply the changes and close the window.

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 display properties of a trace in a 3D report:

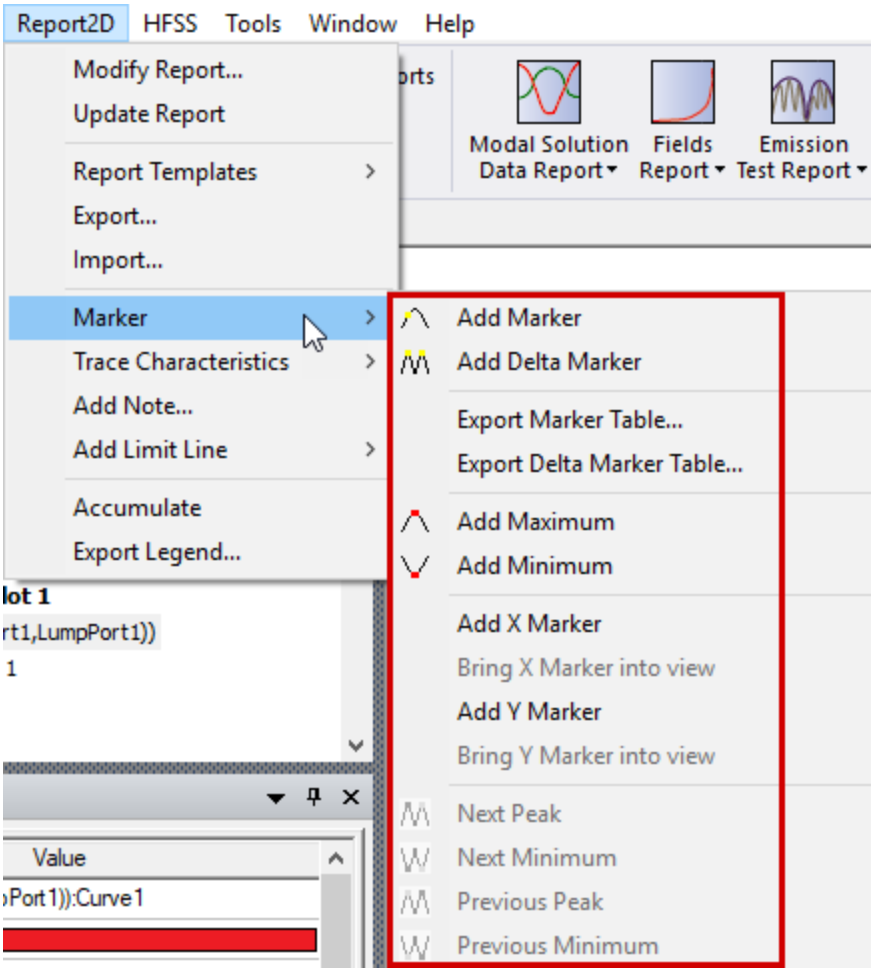
1. Click on the trace. This opens a Properties dialog for the plot with a tab named for the trace selected.
2. The editable properties include Point size, Point Style, whether to show points, whether to show line, line width, and line style.



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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

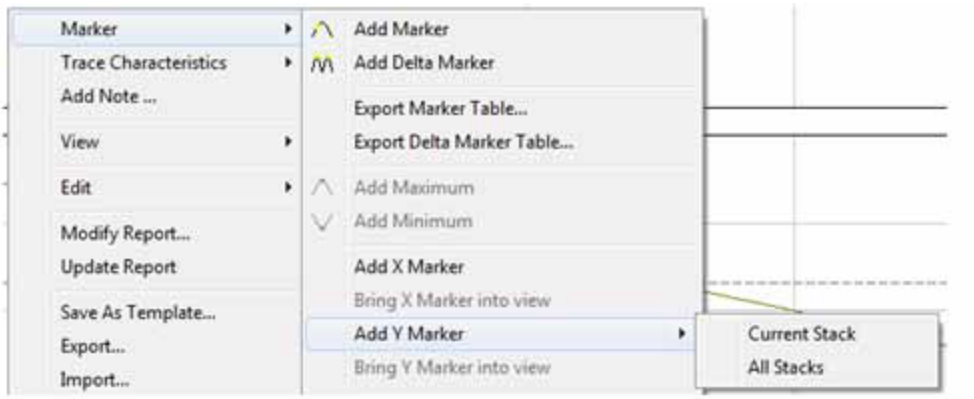
- **Go to Previous** (Left arrow) – moves a selected trace marker to the previous data point.
- **Go to Next** – moves a selected trace marker to the next data point.
- **Go to End** – moves a selected trace marker to the last data point.
- **Next Curve** – selects the next curve in the report, based 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.

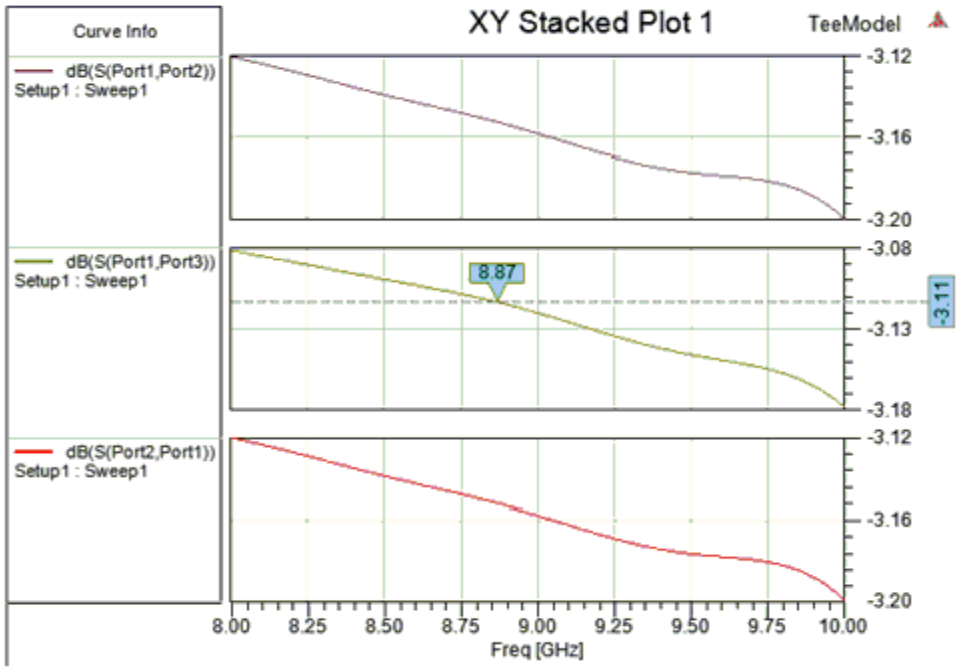
Creating Y Markers in Stacked Plots

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:

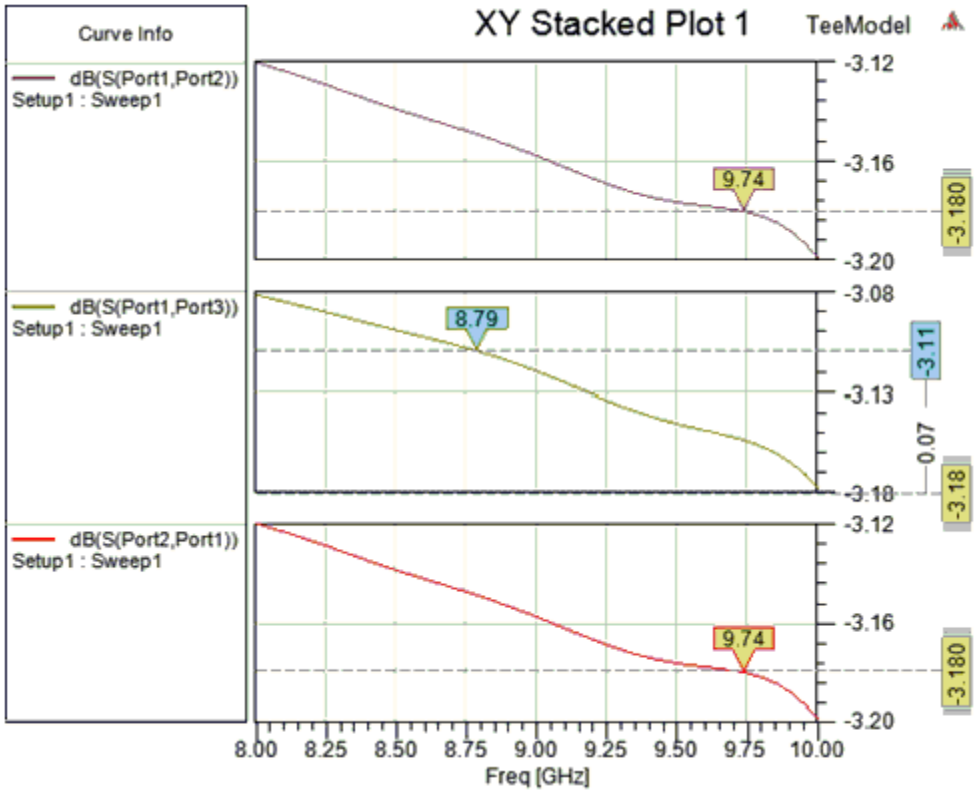


Add Y Marker > **Current Stack** creates a Y marker for the stack on which user performed right mouse button click. The following figure shows that a Y Marker was added to second stack only:

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 > **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:

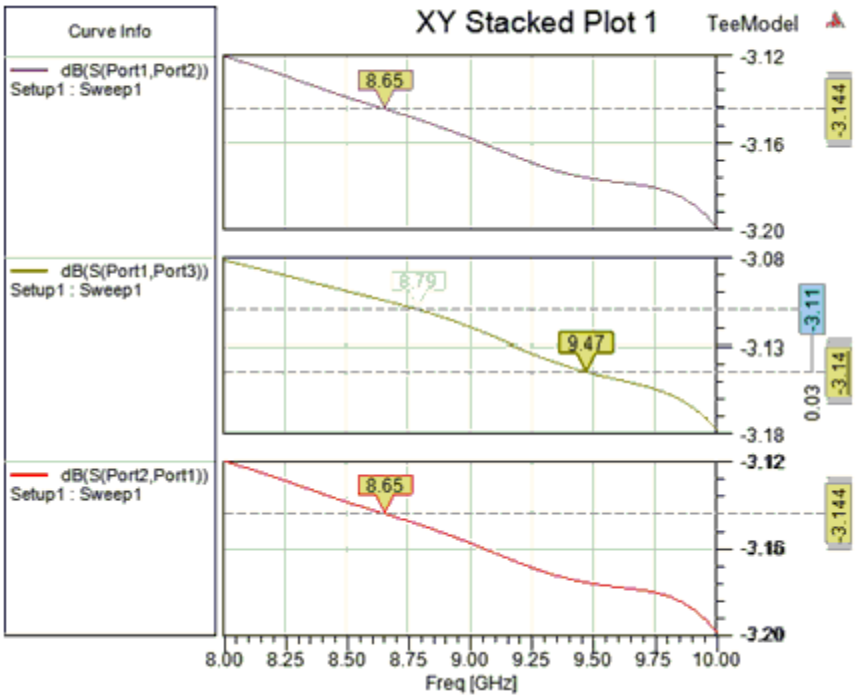


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

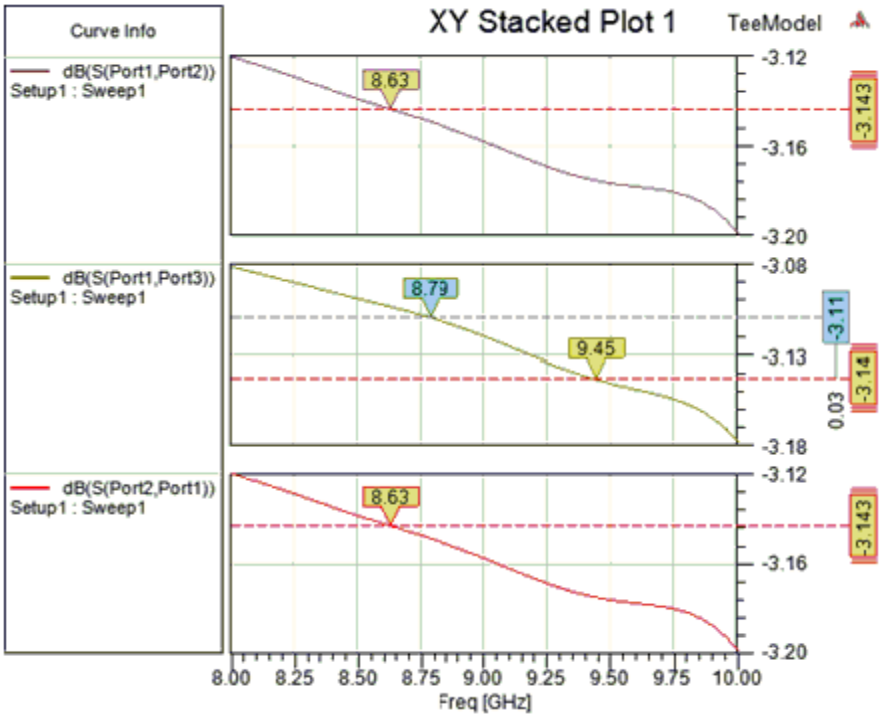
Synchronized Y Markers

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:



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:

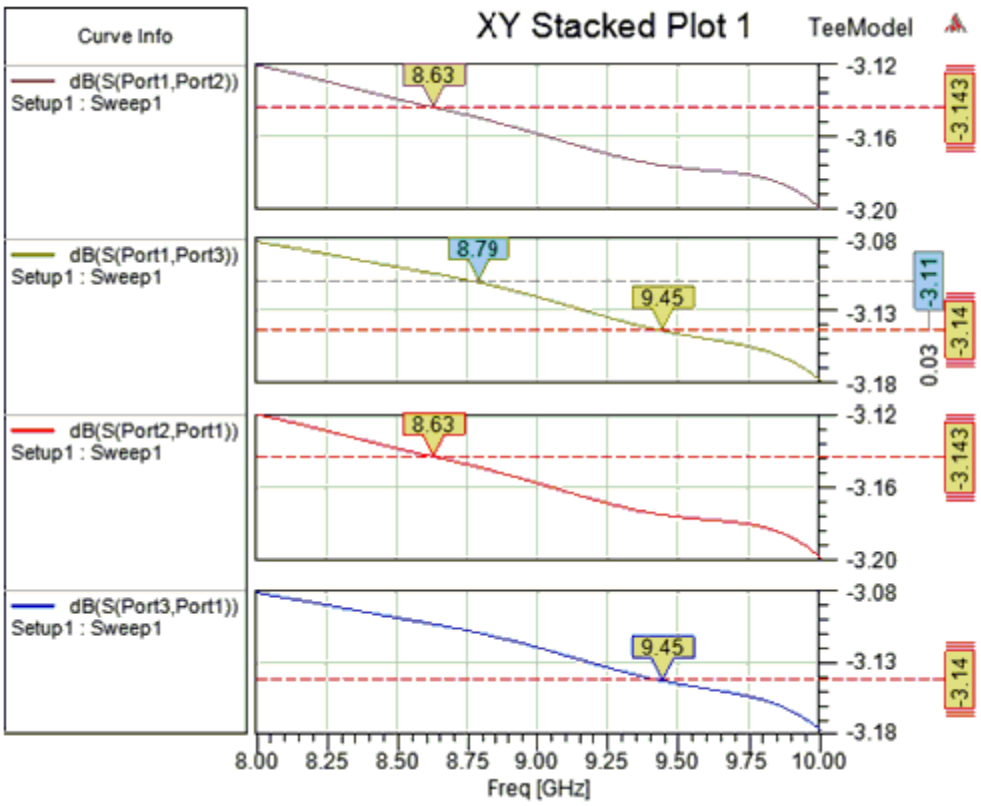
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:

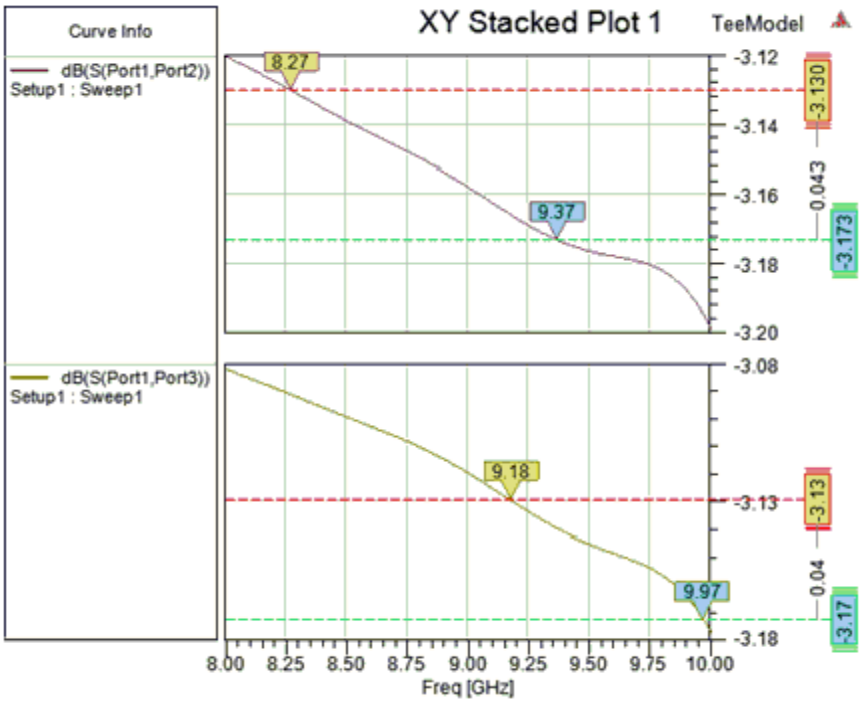
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Y Marker Delta Annotations

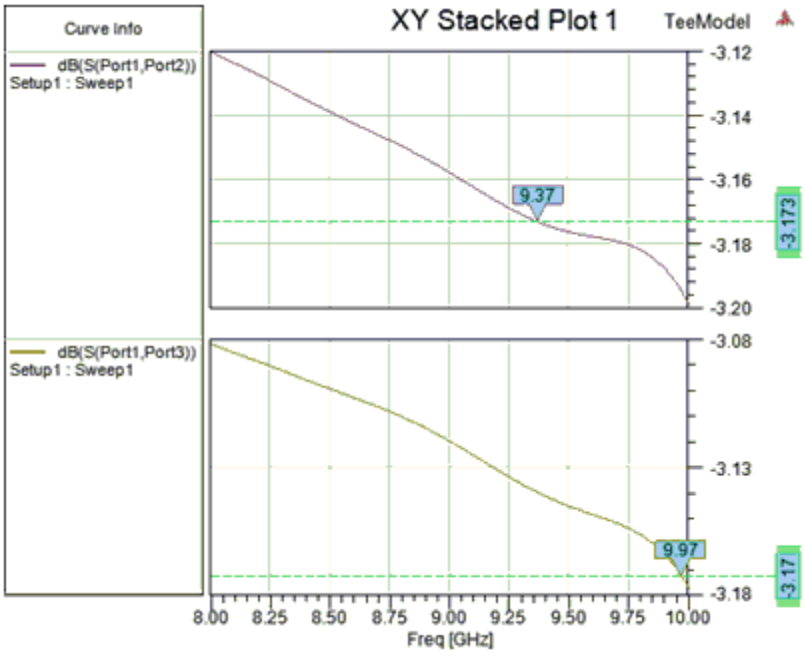
When two more Y Markers are present in a Stacked Eye Diagram, then delta annotations are shown between a pair of adjacent Y Markers in all the stacks, as shown in figure below:

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:

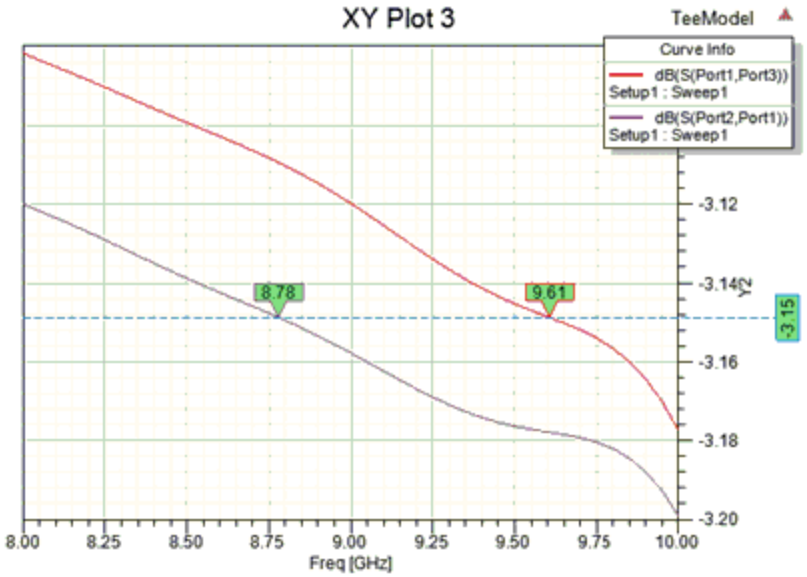


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

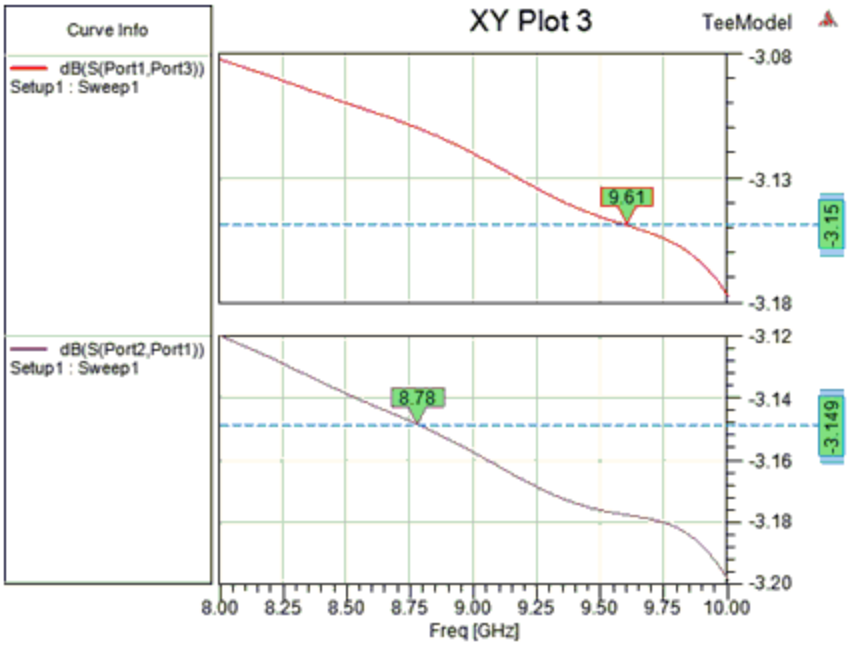
Note that on deleting a stack, Y Markers in other stacks are not affected.

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):



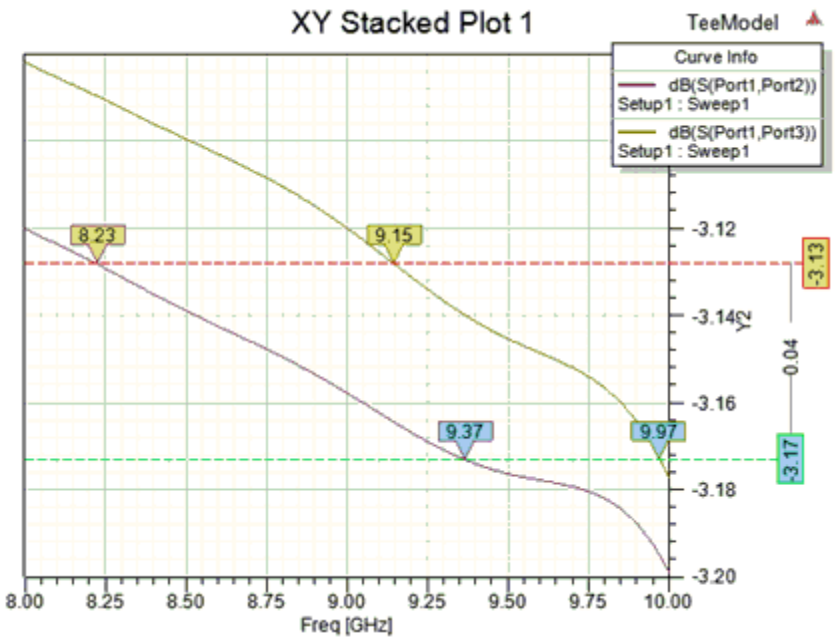
If you change the Display Type property of this plot to Rectangular Stacked Plot then a Rectangular Stacked XY Plot is created with each curve in its own stack and a Y Marker is shown in each stack with value -3.15 (blue Y Marker):



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.

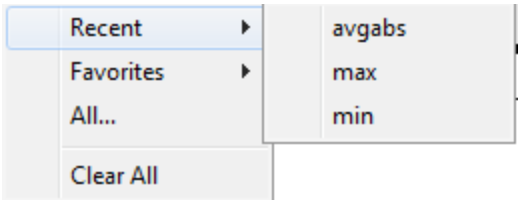
   This opens the **Properties** window for the Axis.

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.

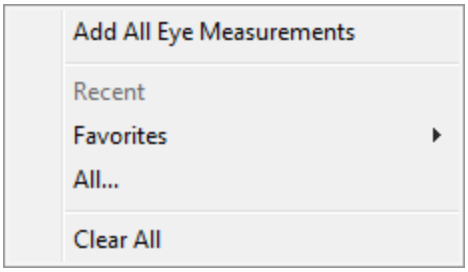
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

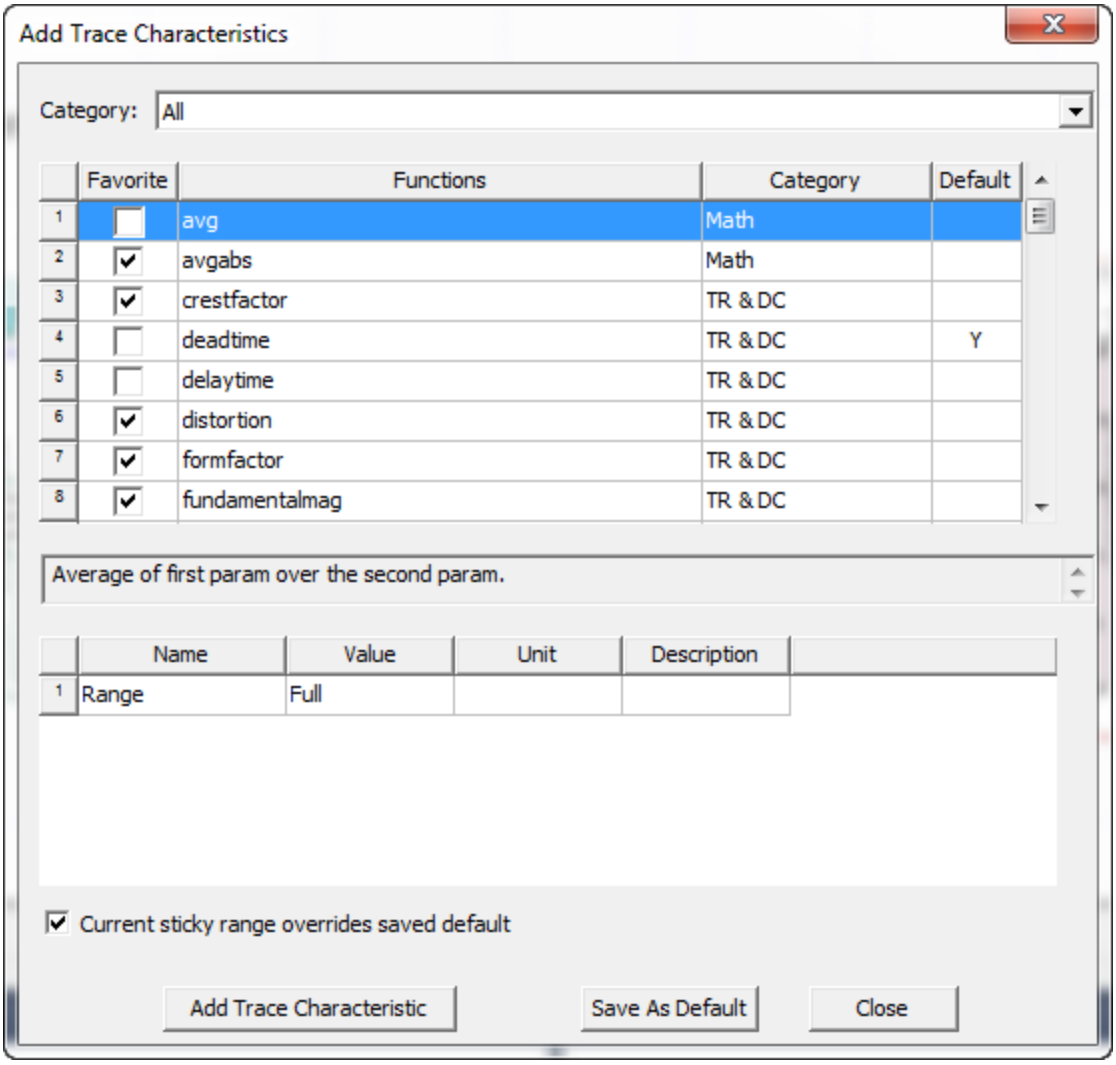
You can add trace characteristics to your list of favorites.

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.

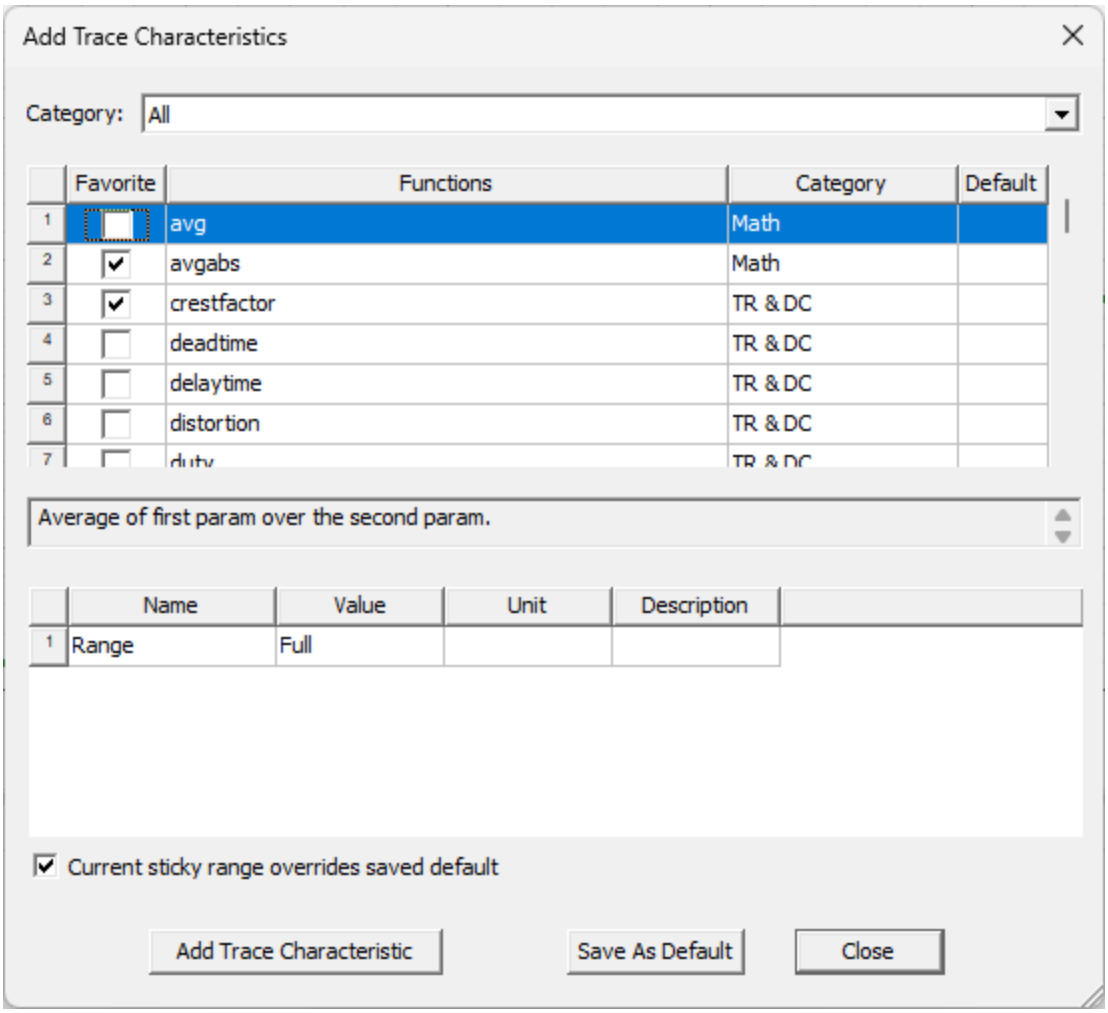
Adding Characteristics Using the Add Trace Characteristics Window

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:

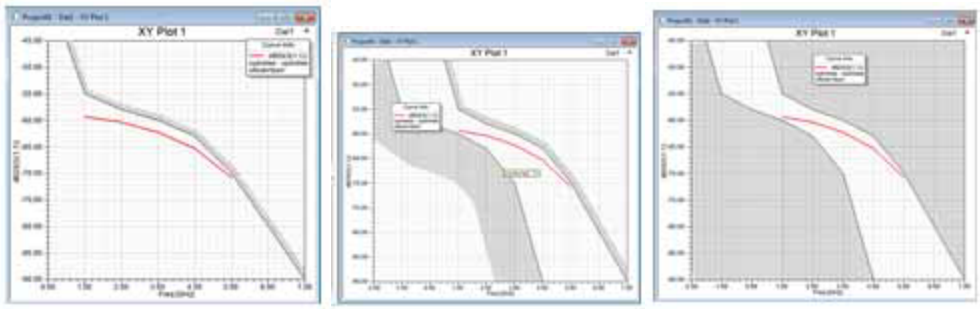
Right-click the report in the target design to which you would like to copy the trace data and select **Paste**.

> **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)



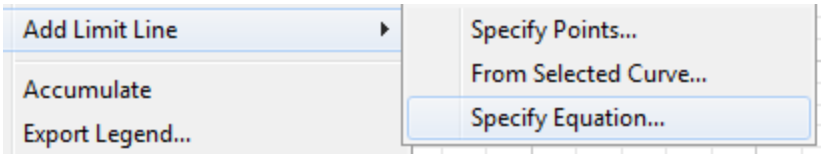
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

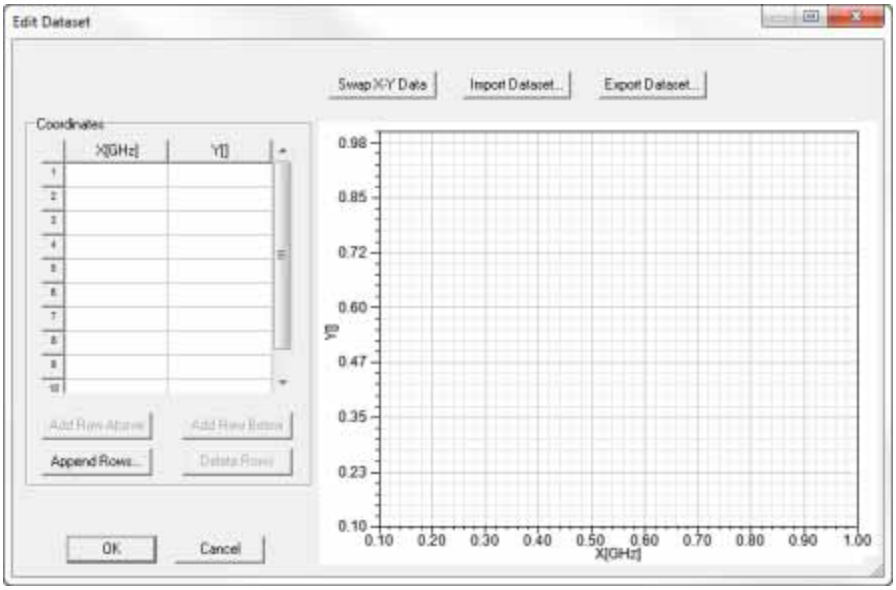
Limit lines are available only on Rectangular (XY) plots, not on XY-like plots such as Bode or Rectangular Stacked. On 2D Plots, the axes extents are based on the extents of the curves.

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**.

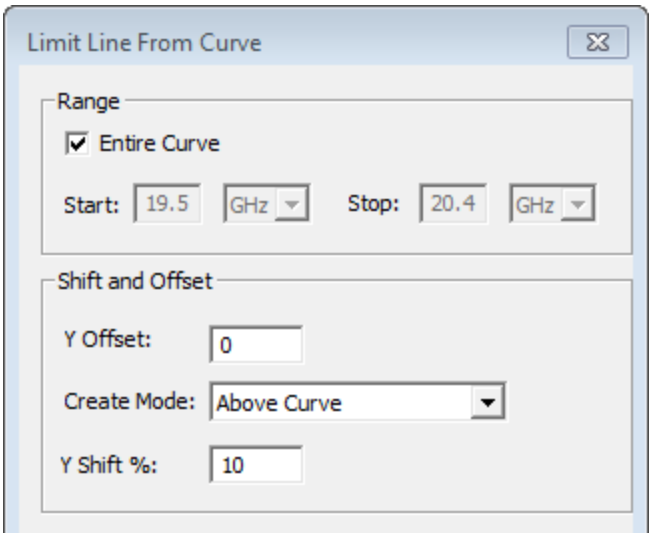


Select **Specify Points** to open an *Edit Dataset* dialog box so that you can specify points.

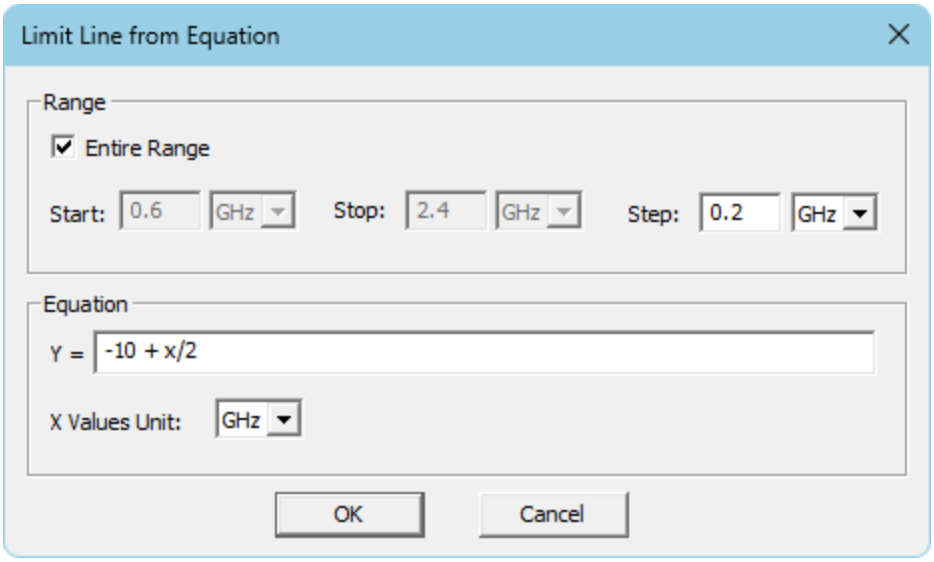


Select **From Selected Curve** to open the *Limit Line From Curve* dialog box.

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.

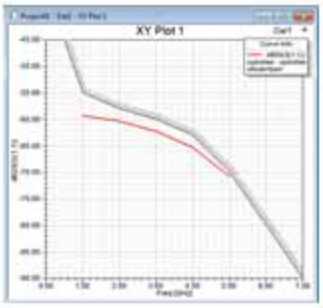
> **Note:**
>
> Each limit line is associated with a particular Y axis (because it has to be scaled the same way as all the curves associated with the axis, follow its log/linear scale and so on). This Y axis association defaults to the first available Y axis when the limit line is created. However, 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.

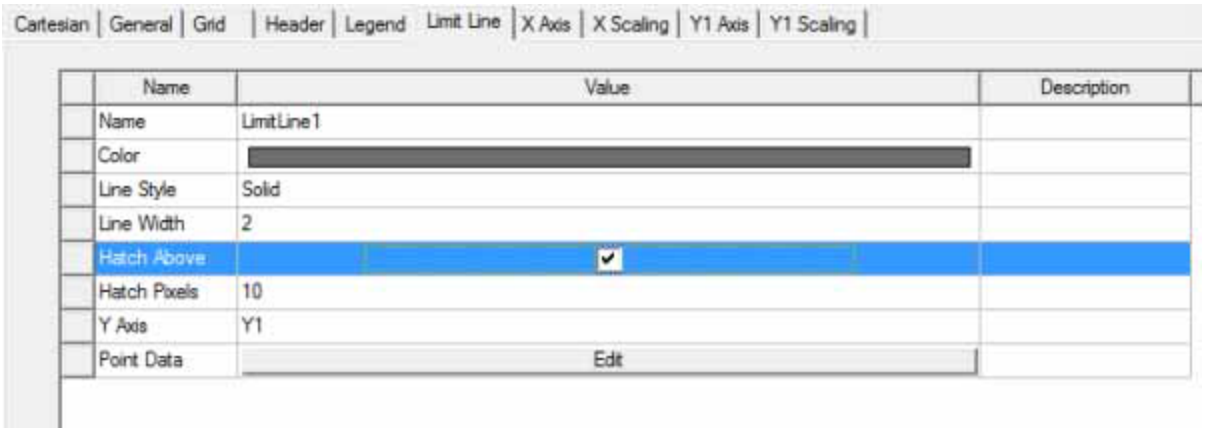
    If you uncheck “Entire Curve,” the Start and Stop fields are enabled and initialized based on the zoom level.

    - 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.



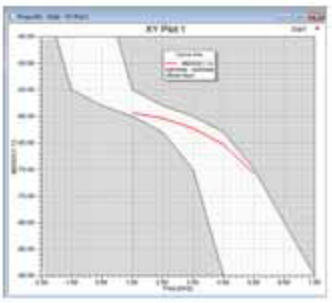
   - Line properties: Color, Style and Width
   - Y axis association
   - Hatch properties
   - Hatch width in pixels
   - Hatch direction (hatch above or below the Limit line)
   - The point definition of the limit line itself.

   For a limit line specified as an Equation, the Properties also include:

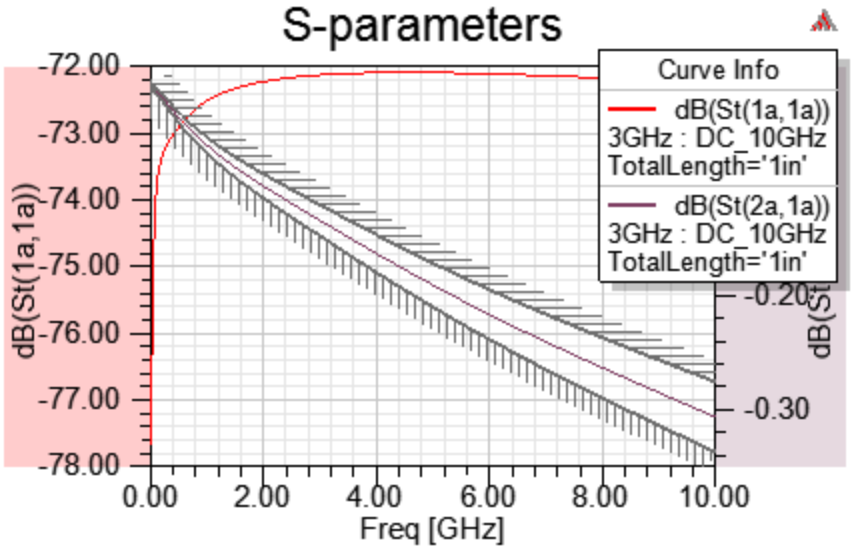
   - Start, Stop, Step, and Equation values.

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.



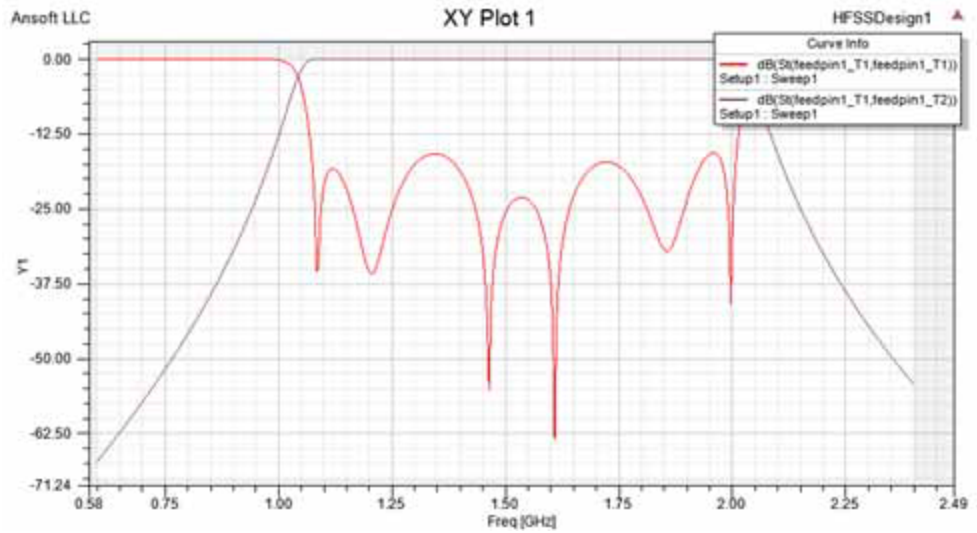
The following example shows the a limit line from curve plot, where the Hatch Above property for the lower limit line has been unchecked.



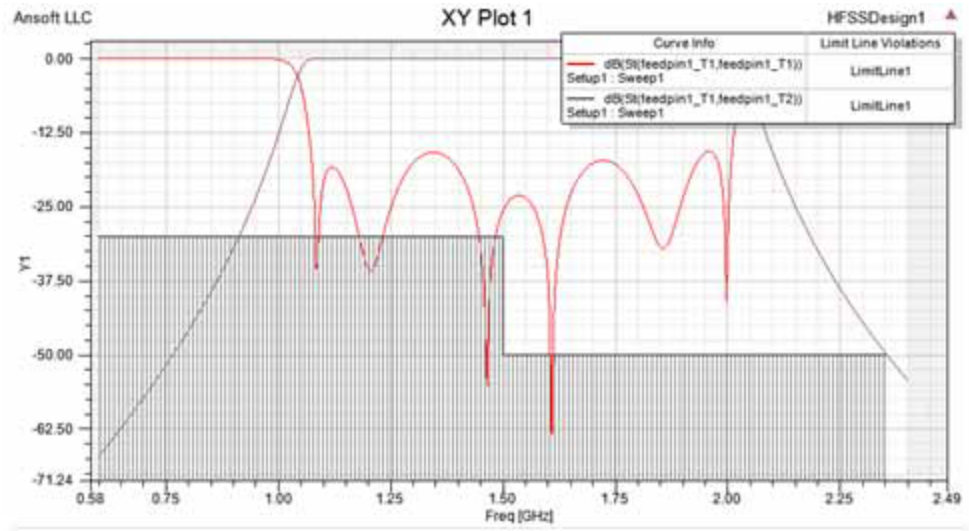
Limit Line Violations

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.

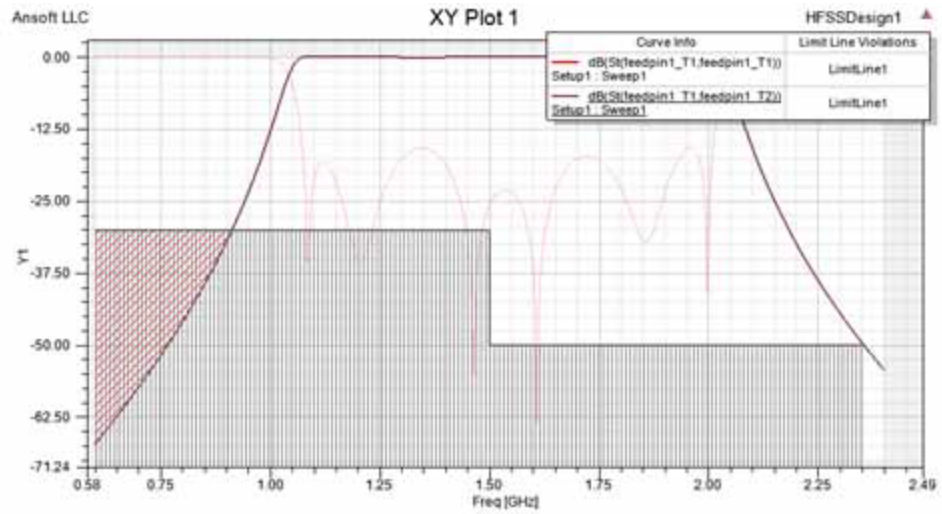


Suppose that the response cannot be below -30 dB until 1.5 GHz, and cannot be below -50 dB at 1.5 GHz or higher. You add a limit line for this requirement in the plot using 'Add a Limit Line' functionality. The plot automatically calculates whether a curve violates this requirement, that is, the limit line and show it in the legends window, as shown in following figure:



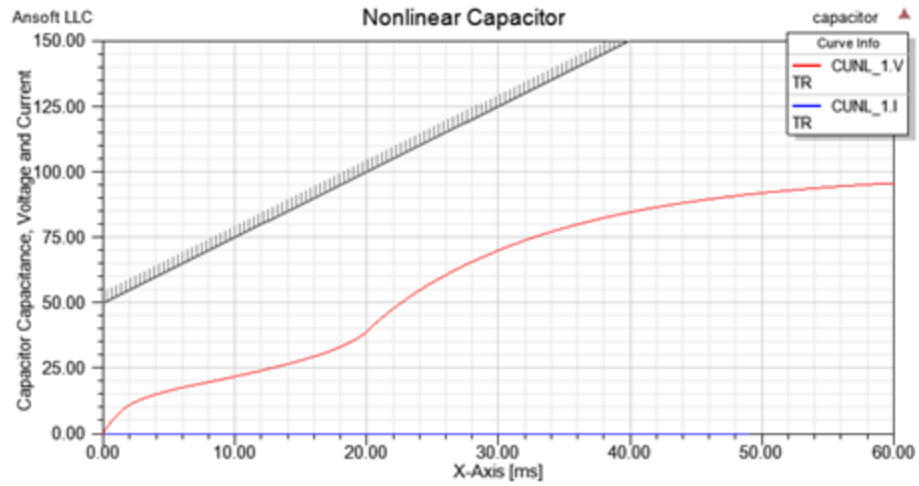
If a curve is selected, then the plot shows the region of the curve that violates the limit line (shaded with slanted red lines), as shown in following figures:

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 HFSS 3D Layout, 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 HFSS 3D Layout 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.

Sweeping Values Across a Sphere

1. Set up a near-field sphere or a far-field infinite sphere.
2. In the **Report** dialog box, click the sphere geometry of interest in the **Geometry** list.
3. For the **Sweeps** variable corresponding to **phi**, select the ellipsis [...] button.

This displays a small dialog.

4. Clear the Use all values check box to enable selection and editing of the sweep values.

All of the possible values for the phi variable are listed in the dialog box. The values are the result of the range of phi you specified during the infinite sphere's setup. To modify the values of phi to be plotted across the sphere, do the following:

a. Click **Edit Sweep**.
b. Specify the following information:

Step or Count	Whether to sweep by steps, or by linear count, decade count, octave count, or exponential count.
Start Value	The point where the rotation of phi begins.
End Value	The point where the rotation of phi ends.
Step or Count	The number of values between the start value and the end value.

c. Click **Update Values**, and then click **OK**.

The values listed are updated to reflect the new number of points.

5. To plot the selected quantity or quantities at every value of phi, select **All Values**.

To plot the selected quantity or quantities at specific values of phi, clear the **All Values** option, and then select the phi values at which you want to plot.

6. For the **Sweeps** variable corresponding to **theta**, follow steps 4 and 5 for modifying the values of theta, if necessary, and specifying the theta values at which to plot the selected quantity or quantities.

> **Note:**
>
> All antenna parameters and maximum far-field data calculated by HFSS is at their maximum over the selected 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 HFSS 3D Layout, a value from a calculated expression, or an intrinsic variable value (such as frequency or theta).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

Available solution quantity categories depend on the design type, solution type, setup parameters, and plot domain. The selected category provides the default name for the plot, such as "S Parameter Plot *n*". You can edit plot names in the project tree and plot header text in the Properties window.

To select solution quantities to plot:

1. Click **Results** > **Standard Report** and select the type of report you would like to generate.

   The **Report** window appears.

2. On the **Trace** tab, select a **Category** from the list.

Category	Description
Variables	Intrinsic variables (such as frequency or theta), or user-defined project variables (such as the length of a quarter-wave transformer).
Output Variables	User-defined expressions applied to derive quantities from the original field solution.
S-Parameter	S-parameters from the S-matrix. For designs which include a Frequency Selective Surface (FSS)-referenced radiation boundary, S_{11} and S_{21} represent the extracted reflection and transmission coefficients, respectively.
Y-Parameter	Admittance matrix parameters computed from the S-parameters and port impedances.
Z-Parameter	Impedance matrix parameters computed from the S-parameters and port impedances.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Group Delay	Quantity calculated as rate of change of the total phase shift with respect to angular frequency: $\frac{d(\Phi)}{d(\omega)}$ Group delay is a measure of the time delay of a signal's envelope as it passes through a system. In the context of scattering (S)-parameters, it tells the user how much of a delay a signal experiences at a specific frequency when traveling through a network (i.e., a filter, transmission line, or an amplifier.) **Group Delay Definition (S-Parameters)** Group delay (τ_g) is defined as the negative derivative of the phase of the S-parameter, with respect to angular frequency (ω): $\tau_g = -\frac{d\emptyset}{d\omega}$ Where: 1. ($\emptyset$) is the phase (in terms of radians) of the relevant S-parameter (i.e. S_{21} for transmission) 2. $\omega = 2\pi f$ is the angular frequency. Alternatively in terms of frequency for (f): $\tau_g = -\frac{1}{2\pi} \bullet \frac{d\emptyset}{df}$ **Example:** If you measure S_{21} over frequency, unwrap the phrase response to compute: $\tau_g(f) = -\frac{1}{2\pi} \bullet \frac{darg(S_{21}(f))}{df}$ This provides a function of frequency that indicates how dispersive the network is (i.e., if different frequency components of a pulse experience different delays).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
VSWR	Voltage standing wave ratio, calculated from the equation: $\frac{1+\lvert S_{ij}\rvert}{1-\lvert S_{ij}\rvert}$
Gamma	Propagation constants for the S-parameters.
Port Z_o	Characteristic port impedances.
Matched S Parameter	Matched S is the S-matrix produced by solver. It becomes useful to view when passive ports exist and the matched S-matrix is rectangular instead of square. When passive ports exist, S is the square matrix of the active ports and the passive ports represent a loss mechanism. Matched S-matrix provides coupling information to the passive ports. In Transient, we show Matched S entries while the simulation is in process which is not clean for regular S when port post processing exists.
Short Circuit Z	Short circuit impedance, calculated as: $\mathrm{ShortCircuitZ}(\mathrm{P1}) = \mathrm{mag}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)$
Loop Inductance	LoopL computes with all other ports shorted, even ports on other nets. These short circuits permit induced currents to flow in nearby nets, meaning LoopL will always be lower than a more rigorous loop inductance computation. $\mathrm{LoopL}(\mathrm{P1}) = \frac{\mathrm{im}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)}{(2 \times \pi \times \mathrm{freq})}$
Loop Resistance	Calculated as: $\mathrm{LoopR}(\mathrm{P1}) = \mathrm{re}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)$
Capacitance	This capacitance calculation leaves other nets open-circuited, whereas a rigorous capacitance calculation would short them to ground, meaning "Capacitance" computed here will always be too low. $\mathrm{C}(\mathrm{P1}) = \frac{\mathrm{im}\left(\frac{1}{\mathrm{Z(P1,\ P1)}}\right)}{(2 \times \pi \times \mathrm{freq})}$
Lambda	Guided wavelength.
Epsilon	Effective permittivity.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
TDR Impedance	Time-Domain Reflectometry (TDR) impedance for non-terminal problems. The idea is to excite a structure with a step function, and inspect the reflections as a function of time. This quantity is available only when the **Time** domain is selected. Selecting the TDR Impedance category displays the TDRZ of every terminal or mode in the ports. The list of available Functions includes those that can operate on the TDRZ values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active S-Parameter	Active S parameters represent the reflection coefficient of one port when another port is excited. HFSS offers passive S-parameters as well as active S-Parameters. If you plot Active S-Parameters, HFSS includes the effects of mutual coupling with other ports or elements. One use is in Optimization, to view how S-parameters change as design parameters change. In this case you may plot Active S Parameters. In such cases they allow you to forecast design behavior in the presence of an ideal feed network. A 0w value is a matched load, but 1w values take that into account. Active S-parameters are given as a linear combination of the regular S-parameters, with some weights 'a', based on the matrix equation $\mathbf{b} = \mathbf{S} \times \mathbf{a}$. In the post-processor, the active S-parameter for port*n* is displayed to the user as ActiveS(P*n*:*m*) where "P*n*" indicates the port number and ":*m*" indicates the mode number. Active S, Y, Z, VSWR are supported only for driven modal projects. Power is proportional to the square of the fields. As a consequence, active S-parameters, which are field related, are proportional to the square root of what you set under Edit Sources. You can check this by setting all power to 0 except for one port of interest where you want to see the Active S-parameter. Then for a neighboring port, change the input power in steps, and monitor how the active S-parameter changes. Keep in mind that this involves complex numbers which complicates things. Given a driven model project with a total of *N*port modes, let a_k denote the complex excitation for the *k*-th mode specified in the Edit Sources dialog box. Also let S denote the computed *N* x *N*scattering matrix. If $a_m \neq 0$ define active-S_m, *m* = 1..., *N* by $$active\text{-}S_m \equiv \sum_{n=1}^{N} S_{mn} \frac{a_n}{a_m} \qquad m = 1, \ldots, N$$ If *S* is renormalized or deembedded, the interpretation is that the stimulations are applied to the renormalized external transmission lines at the plane of deembedding. The other relative active quantities are simply transformations on the active S_m.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active Y-Parameter	$Y_0^{(M)}$ is the port admittance of the *m*-th mode. $active\text{-}Y_m \equiv Y_0{}^{(m)} \frac{1 - active\text{-}S_m}{1 + active\text{-}S_m} \qquad m = 1, ..., N$
Active Z-Parameter	$Z_0^{(m)}$ is the impedance of the *m*-th mode. $active\text{-}Z_m \equiv Z_0{}^{(m)} \frac{1 + active\text{-}S_m}{1 - active\text{-}S_m} \qquad m = 1, ..., N$
Active VSWR	Active Voltage standing wave ratio is supported only for driven modal projects. $active\text{-}VSWR_m \equiv \frac{1 + \lvert active\text{-}S_m \rvert}{1 - \lvert active\text{-}S_m \rvert} \qquad m = 1, ..., N$
Passivity	A scalar quantity based on the matrix Q = I - conjugate (transpose(S)) * S For every frequency, the value must be no larger than 1. A uniform renormalization of 50 ohms is performed on the solution data for Passivity checking.
Power	For Composite Excitation Solution types, selecting Power as Category lists Quantities for Accepted (Pacc), Reflected (Pref), and Incident (Pinc) powers/port based on your Edit Sources settings.
Design	Enables you to plot or tabulate properties of objects, such as their volumes.
Expression Cache	The values of expressions listed in the Expression cache of the Solution setup can be plotted, for example, as a function of adaptive pass to monitor their convergence.
Expression Convergence	This is intended to plot convergence, as a function of adaptive pass, of expressions in the Expression cache of the Solution setup. In defining the report, for "Context", select Solution: SetupN:Adaptive Pass. ExprDelta will show the change in the value of the expression as a function of adaptive pass, while ExprGoal will show, for comparison, the convergence goal for this expression, as defined in the Analysis Setup under the panel Expression Cache.

3. Select a **Quantity** from the list.
4. If desired, select a **Function**.
5. Click **Close**.

Selecting a Field Quantity to Plot

When plotting field quantities, the quantity can be a value that was automatically calculated by HFSS 3D Layout such as the magnitude of S_{11}, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or phase.

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 field quantity to plot:

1. When you create the report, specify the Report Type as "Fields" and the plot type (for example, radiation pattern.)
2. In the *Report* dialog box, select Geometry for the Context, unless you are plotting scalar (for example, integration).
3. In the *Report* dialog box, select one of the following categories. The selected category provides the default plot name. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Variables	Intrinsic variables, such as frequency or phase, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	User defined expressions applied to derive quantities from the original field solution.
Calculator Expressions	Includes scalar and vector field quantities automatically calculated by HFSS 3D Layout, 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.

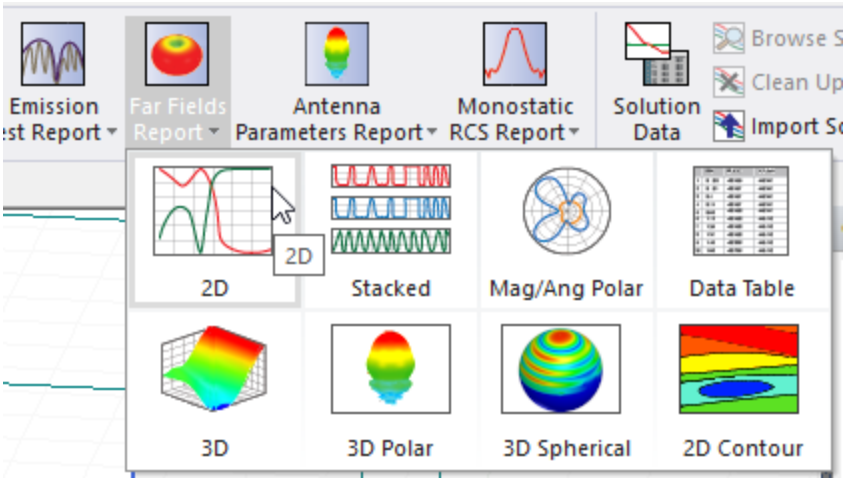
Selecting a Far-Field Quantity to Plot

When plotting far-field quantities, the quantity can be a value that was calculated by HFSS such as antenna gain, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.

Prerequisites

For the **Far Field Report** icons to appear on the ribbon, or for the **Create Far Field Reports** command to appear on the **Results** menus, the design must contain radiation boundaries, PMLs or Hybrid regions, and must contain a Far 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.

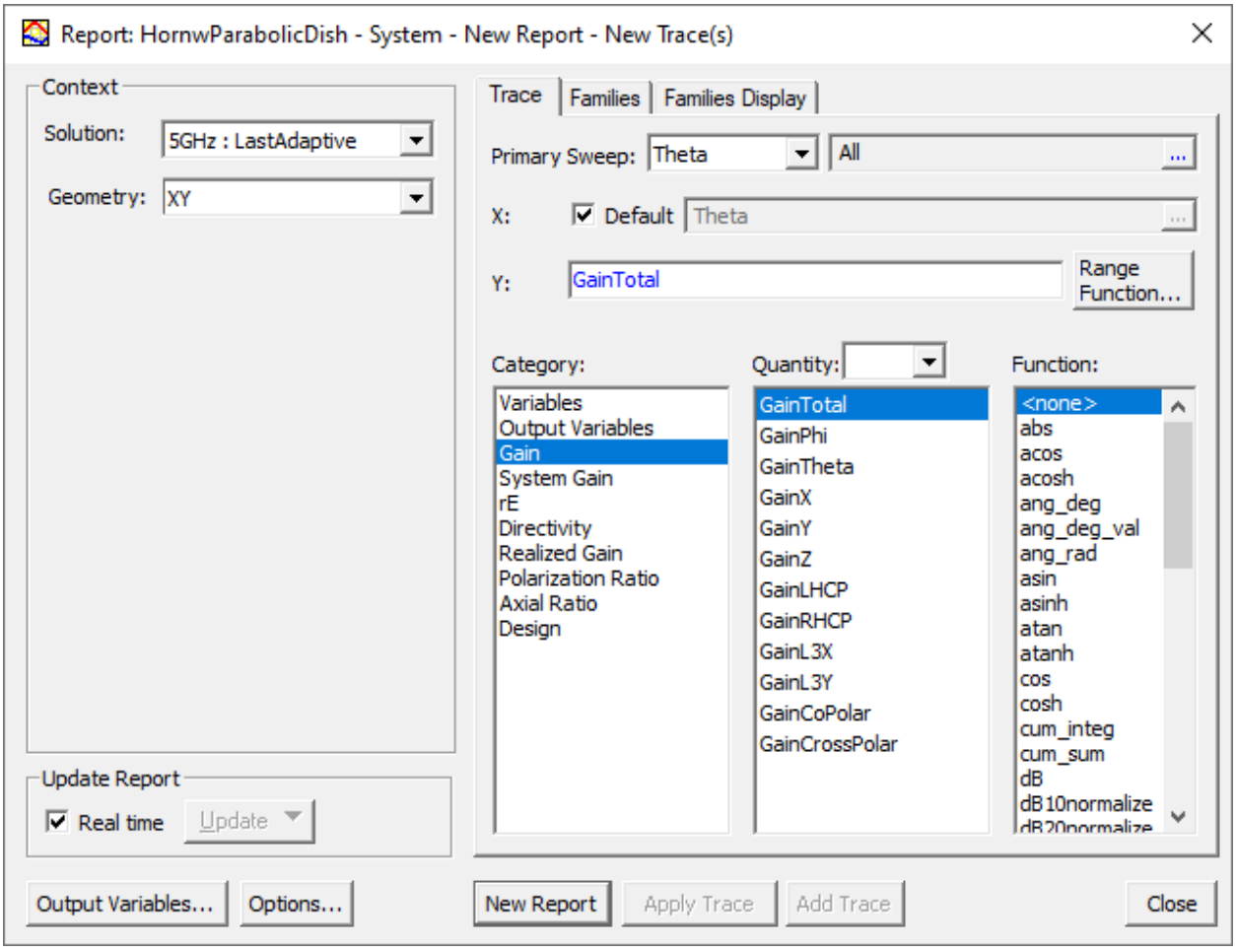


To tune Far Field plots based on partial quantities, the design must include a variable from which derivatives can be calculated. See: Derivative Tuning for Reports.

To plot a far-field quantity:

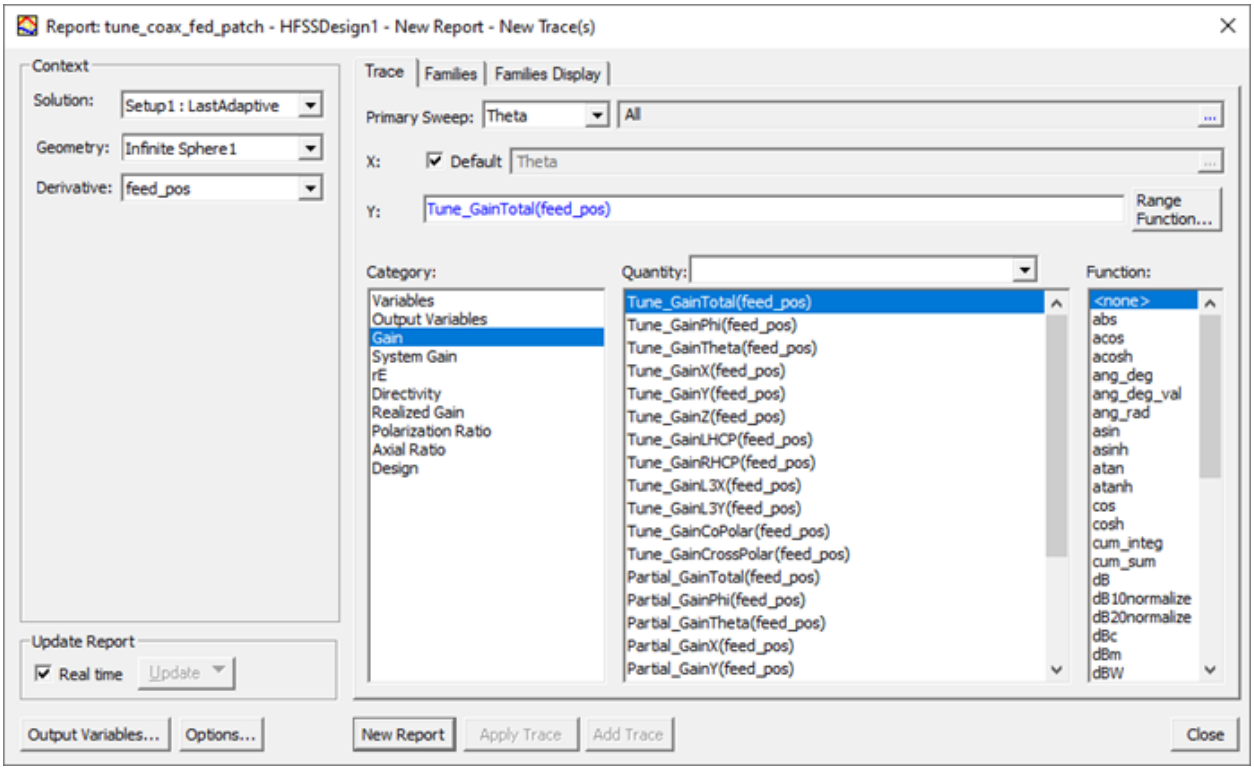
1. From the **Results** ribbon, select the **Far Fields Report** icon, or select **Results** > **Create Far Fields Report** to open the **Report** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If your design includes one or more variables for derivatives, you can select one or all derivatives. Selecting a **Category** will list tunable **Quantities** available:

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 **Context** area, select the **Solution**, **Geometry**, and **Source**.

> **Note:**
>
> In the case of finite array, select a Source group.

3. Make a selection from the **Category** list. The available options may differ depending on previous choices. Selecting one or more Derivatives results in Tunable variations of these quantities being listed:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

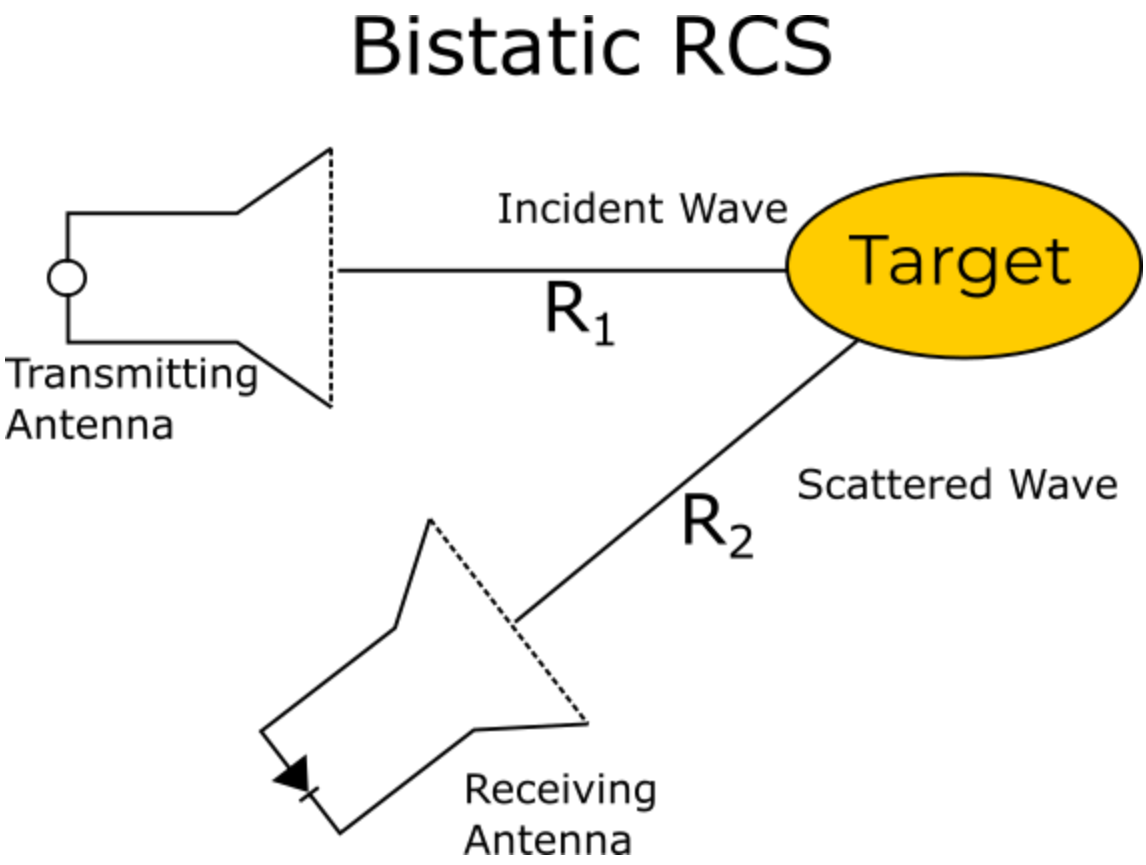
Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	User-defined expressions applied to derive quantities from the original field solution.
Gain	Gain is 4π times the ratio of an antenna's radiation intensity in a given direction to the total power accepted by the antenna.
System Gain	System gain appears in the Quantity list if Edit Sources was used to specify system power for gain calculations. System gain is 4π times the ratio of an antenna's radiation intensity in a given direction to the user-specified power.
rE	The selected component of the radiated electric field, which is multiplied by the radial distance, *r*.
Directivity	Directivity of the antenna.
Realized Gain	Realized gain is 4π times the ratio of an antenna's radiation intensity in a given direction to the total power incident upon the antenna port(s).
Polarization Ratio	Polarization ratio of the electric field.
Axial Ratio	Axial ratio of the electric field.
Normalized Bistatic RCS	The normalized radar cross-section: $\frac{\sigma}{\lambda_0^2}$ where $\lambda_0$is the wavelength of free space. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.*
Radar Cross-Section (Bistatic RCS)	The radar cross-section (RCS) or echo area (σ) is measured in meters squared and represented for a bistatic arrangement (e.g., when the transmitter and receiver are in different locations as shown in the figure below). This is represented by: $\sigma = \frac{4\pi r^2 \lvert E_{\text{scat}} \rvert^2}{\lvert E_{\text{inc}} \rvert^2}$ where:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	• E_{scat} is the scattered E-field. • E_{inc} is the incident E-field. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.*
Complex (Bistatic) RCS	The equation for complex (bistatic) RCS is calculated as: $\sqrt{\sigma} = 2\sqrt{\pi \mathrm{R}}\frac{E_{scat}}{\lvert E_{inc}\rvert}$ where: • E_{scat} is the scattered E-field. • E_{inc} is the incident E-field. This form retains the phase information. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.*
Monostatic RCS	The radar cross-section (RCS) or echo area when the transmitter and receiver are at the same location. For Monostatic RCS, you need not be concerned with the Theta and Phi values defined in the radiation sphere. Only the incident wave Theta and incident wave Phi are used in calculating a Monostatic RCS plot. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.* *A proper incident angle sweep should exist at the incident wave source setup before HFSS can plot Monostatic RCS*

The following diagram shows the bistatic RCS concept, with separate transmitting and receiving antennas.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each **Category** item selected updates the **Quantity** list to offer quantities appropriate to selected category. **Category** selection for a Variable of an Output Variable lists those available in each case. Selecting Antenna Parameters as Category causes the Quantity list to show Antenna parameters. The selected Category provides the default report name, such as rE Plot *n*.

4. Select the **Quantity** to apply to the selected **Category**.

   If the **Category** item is rE, Gain, Directivity, or Realized Gain, you will need to specify the polarization of the electric field by selecting from the **Quantity** list. If you manually enter Phi or Theta component of a far field quantity when using Az/El or El/Az far field definition, the Report uses the Z axis of the Az/El or El/Az coordinate system definition as pole to calculate theta/phi component for rE.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total	The combined magnitude of the electric field components.
Phi	The phi component (φ).
Theta	The theta component (θ).
X	The x component.
Y	The y component.
Z	The z component.
LHCP	The dominant component for a left-hand, circularly polarized field.
RHCP	The dominant component for a right-hand, circularly polarized field.
CircularLHCP	The polarization ratio for a predominantly left-hand, circularly polarized antenna.
CircularRHCP	The polarization ratio for a predominantly right-hand, circularly polarized antenna.
SphericalPhi	The polarization ratio for a predominantly φ-polarized antenna.
SphericalTheta	The polarization ratio for a predominantly θ-polarized antenna.
L3X	The dominant component for an x-polarized aperture using Ludwig's third definition of cross polarization.
L3Y	The dominant component for a y-polarized aperture using Ludwig's third definition of cross polarization.
CoPolar	Reports fixed polarization rotated around Z axis by “slant” angle from X axis of coordinate system of Infinite sphere definition: $rE_{CoPolar} = \cos(\varphi - a)\hat{\theta} - \sin(\varphi - a)\hat{\varphi}$ where *a* is the slanted angle.
CrossPolar	$rE_{CrossPolar} = -\sin(\varphi - a)\hat{\theta} - \cos(\varphi - a)\hat{\varphi}$ where *a* is the slanted angle.

The plot's Y axis field shows the combined selections.

For example, if you select Gain as the Category and RHCP as the Quantity, HFSS evaluates the equation as:

$$Gain_{RHCP} = \frac{4\pi U_{RHCP}(\theta,\varphi)}{P_{acc}} = \frac{4\pi |rE_{RHCP}(\theta,\varphi)|^2}{2\eta P_{acc}}$$

$$= \frac{4\pi \left| r\vec{E}(\theta,\varphi) \bullet \left(\frac{\hat{\theta} - j\hat{\varphi}}{\sqrt{2}} \right) \right|^2}{2\eta P_{acc}}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. You can also select a function to apply to the your selections for the **Category** and **Quantity** (e.g., mag).

   As you make selections for **Category**, **Quantity**, and **Function**, the **Y** field shows the combined calculation they describe.

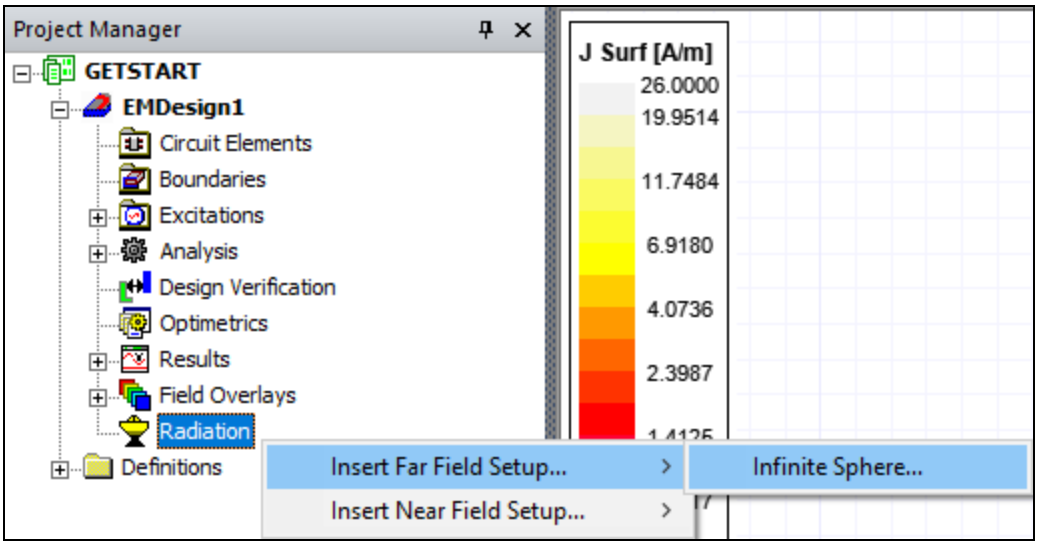
6. When you are finished, click **New Report** to create the report.

   The new report based on your selections is displayed.

Set Up a Far Field Infinite Sphere

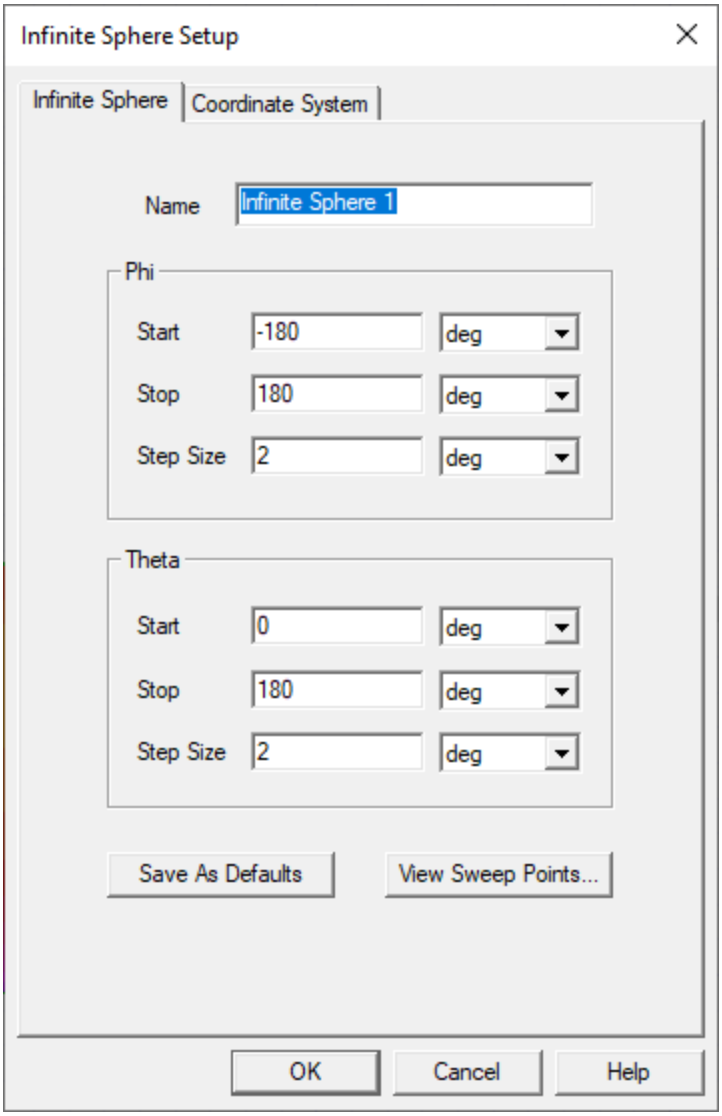
Follow these steps to set up a far field infinite sphere and plot the results.

1. From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]** (i.e., EMDesign1). Then right-click **Radiation** and select **Insert Far Field Setup** > **Infinite Sphere**.



   The **Infinite Sphere Setup** window opens.

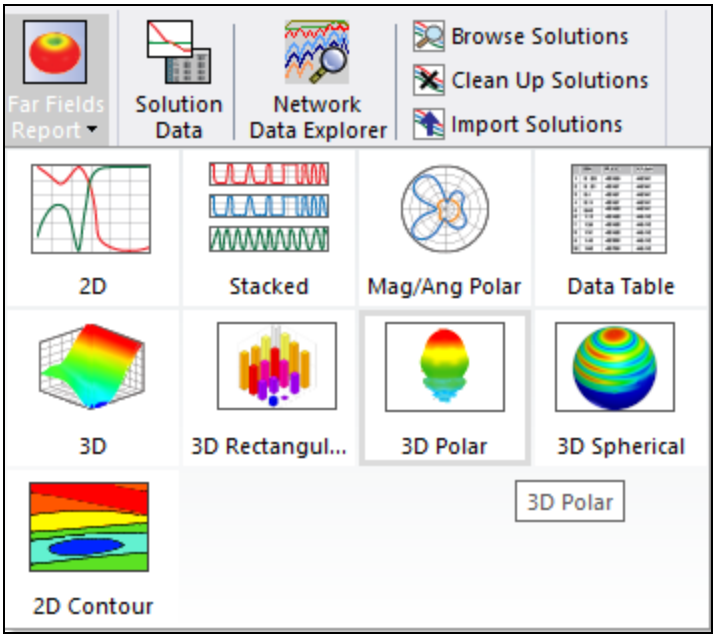
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Make any appropriate changes to the default configuration and click **OK** to close the **Infinite Sphere** window and add the far field setup to the design.

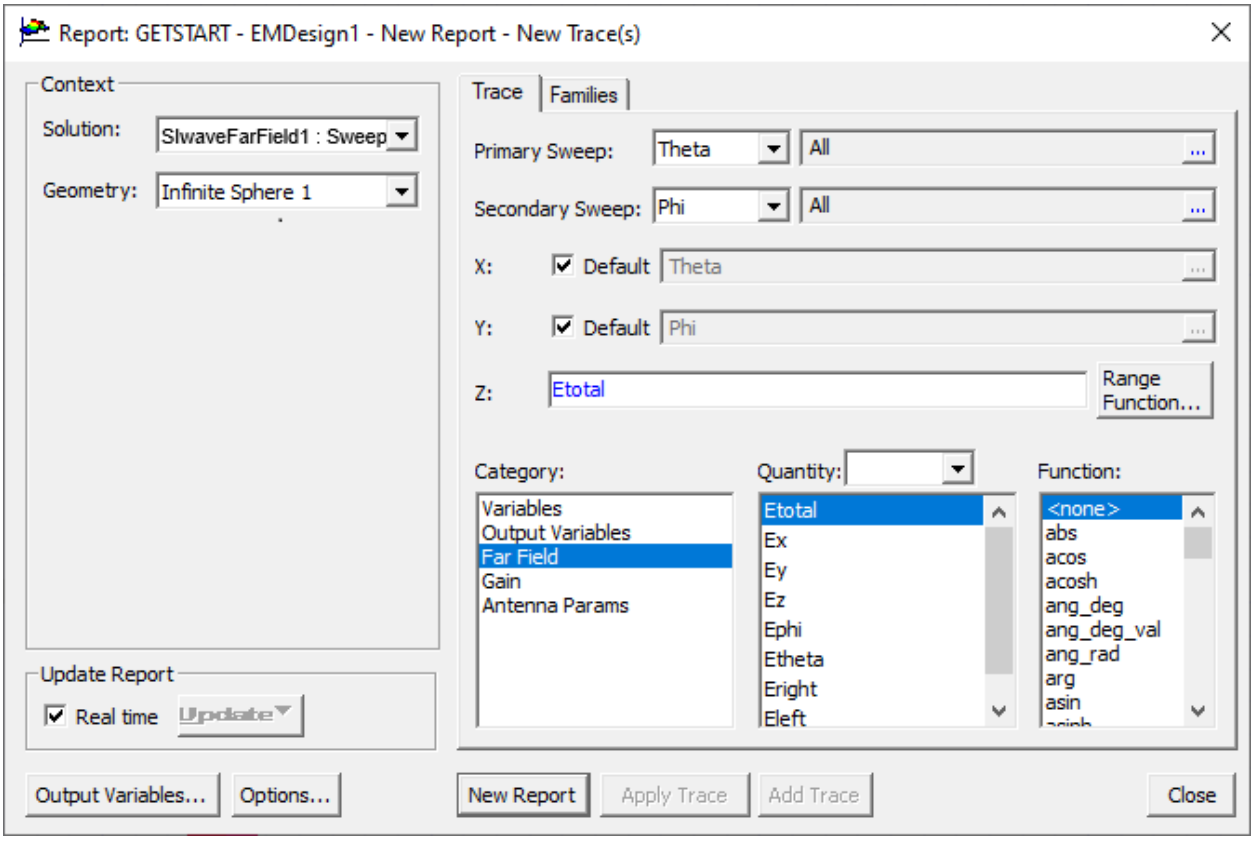
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the **Results** ribbon, select a report type (i.e., **Far Fields Report** > **3D Polar**).



The **Report** 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.



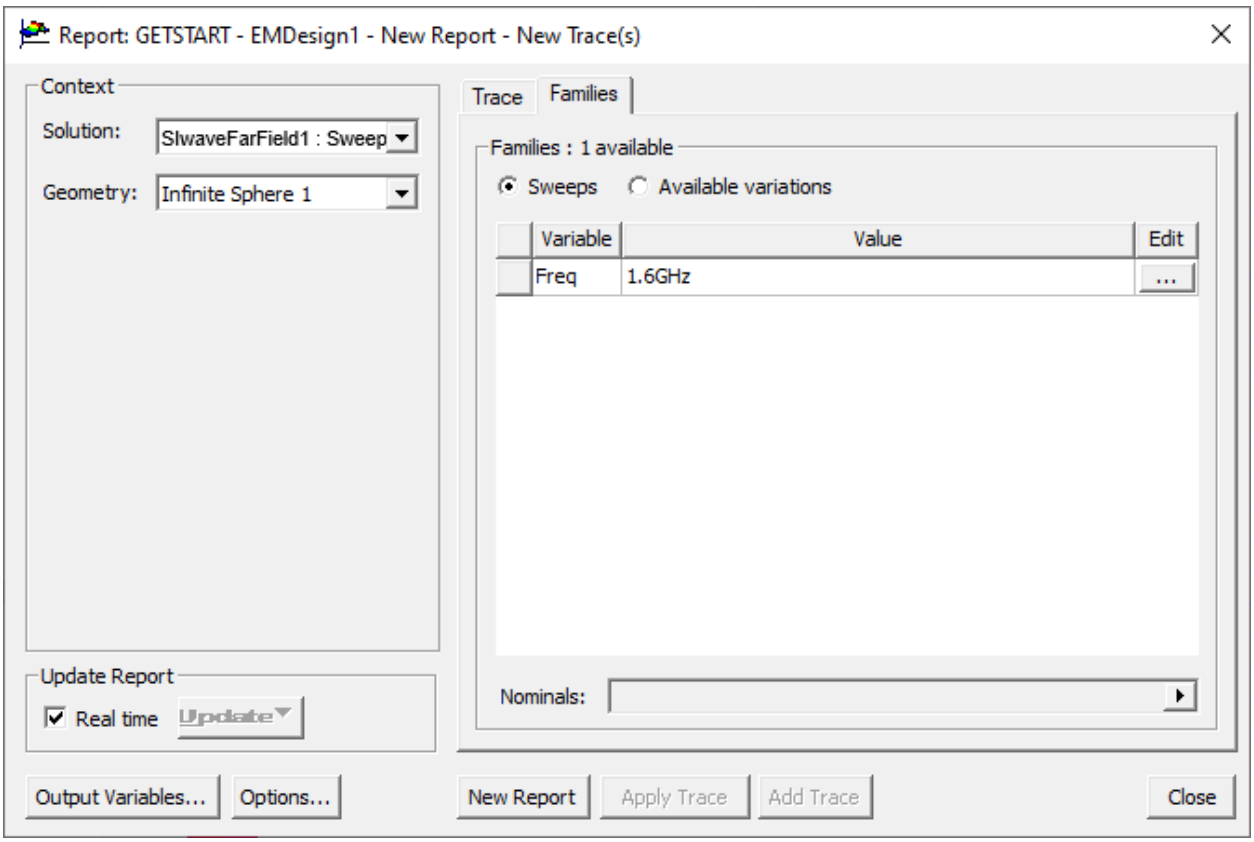
Note: Alternatively, open the **Report** window by doing one of the following:

- From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Results** and select a report type (i.e., **Create Far Fields Report** > **3D Polar Plot**).
- From **HFSS 3D Layout**, select **Results** > **Create Far Fields Report** > **[*Report Type*]** (i.e., 3D Polar Plot).

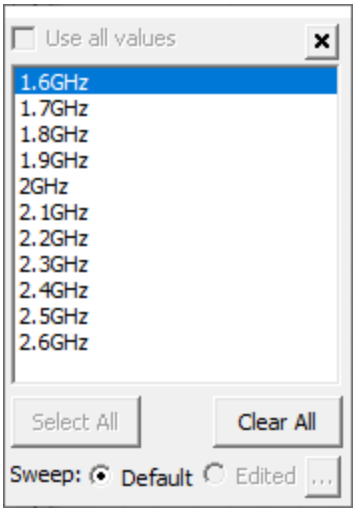
Note: In Step 4, the **Solution** and **Geometry** drop-down menus pre-select available choices that provide far field results (i.e., **SIwaveFarFieldSetup: Sweep2** is selected in the **Solution** drop-down menu, and **Infinite Sphere 1** is selected in the **Geometry** drop-down menu) and the lists in the **Trace** tab pre-select the appropriate choices (i.e., **Far Field** is selected in the **Category** list, **Etotal** is selected in the **Quantity** list, and **<none>** is selected in the **Function** list).

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 **Families** tab. From the **Families** tab > **Freq** row, click the **[...]** button in the **Edit** column.

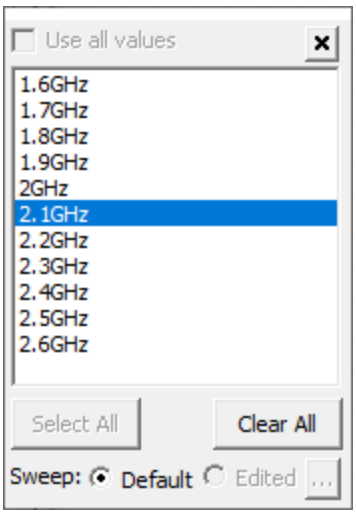


A value list opens.



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 an appropriate value (i.e., in this example, **2.1GHz** corresponds closest to the point of minimum return loss). Then click outside of the value list to close it.

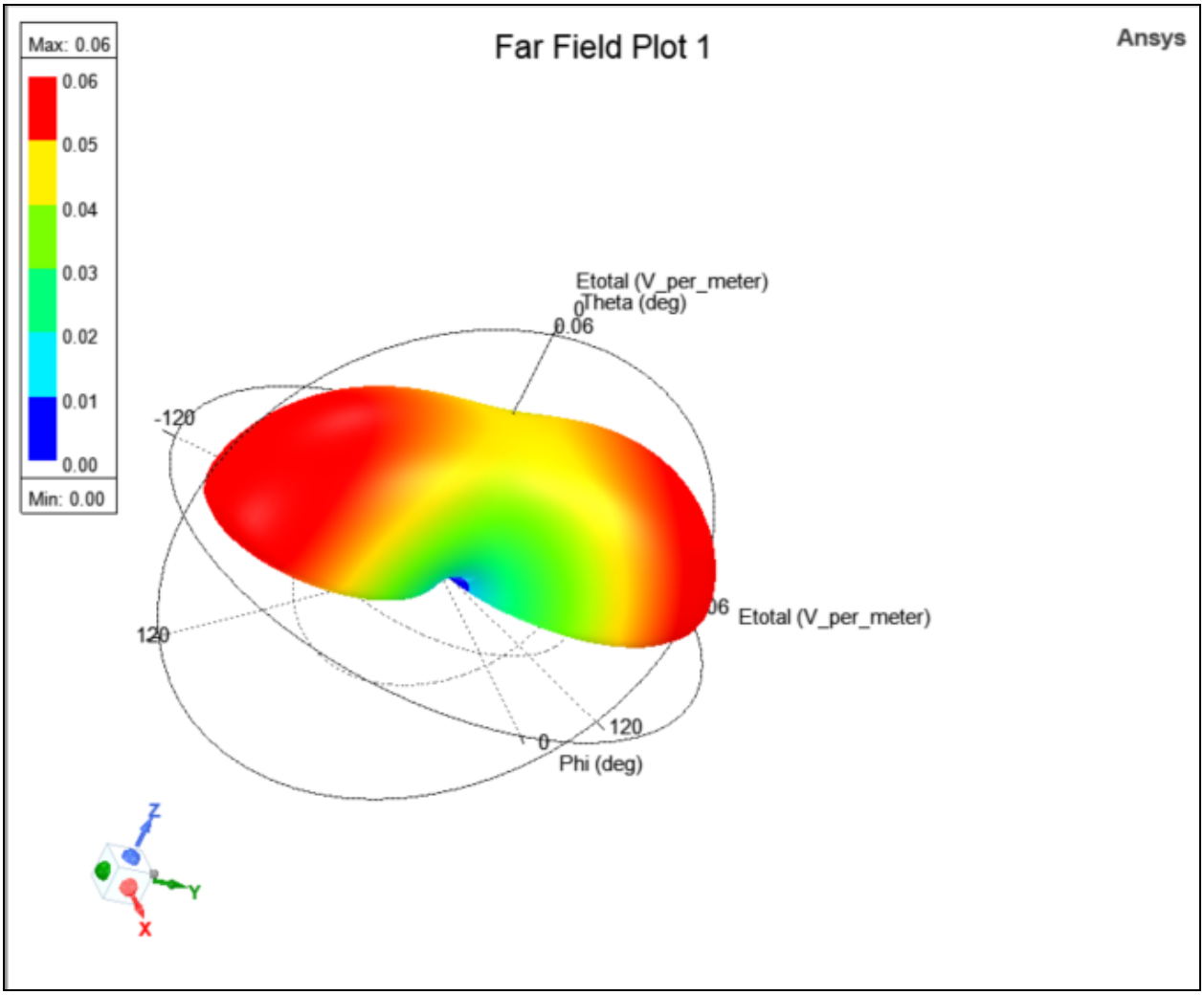


6. From the **Report** window, click **New Report** and the far field plot opens under the **Report** window. **Close** the **Report** window to access the plot.
7. **Close** the **Report** window to view the plot.

> **Note:** Alternatively, open the **Report** window by doing one of the following:
>
> - From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Results** and select **Create Far Fields Report** > **[*Report Type*]** (i.e., 3D Polar Plot).
> - From **HFSS 3D Layout**, select **Results** > **Create Far Fields Report** > **[*Report Type*]** (i.e., 3D Polar Plot).

8. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** the far field plot using the standard **Layout Editor** controls.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Plotting Vertical Cross-Sections of Far Fields

When plotting far fields, a vertical cross-section plot results from holding phi fixed and sweeping theta through a range of values.

1. Open the **Report** dialog box.
2. Click the ellipsis **[...]** button for the **sweep variable corresponding to phi.**

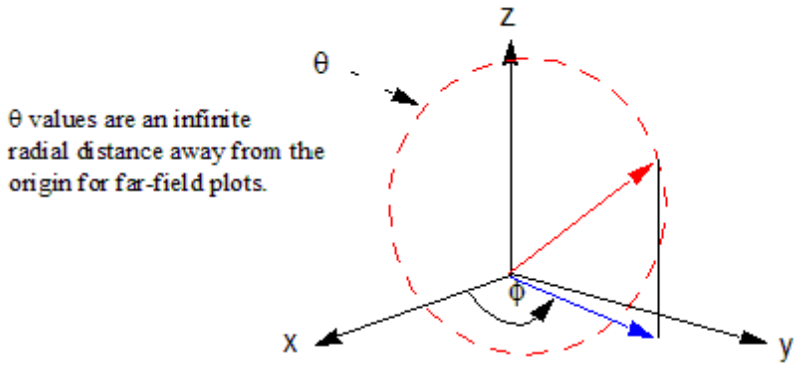
   This displays a dialog listing all values for the phi variable. The values are the result of the range of phi you specified during the infinite sphere's setup.

3. Select the fixed value that phi should take in the plot.

   HFSS will display values for the vertical cross-section at selected phi cuts of the problem region at a set of theta rotations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The figure shown below demonstrates the orientation of the vertical cross-section when ϕ is the fixed variable:



Plotting Horizontal Cross-Sections of Far Fields

When plotting far fields, a horizontal cross-section results from holding theta fixed and sweeping phi through a range of values.

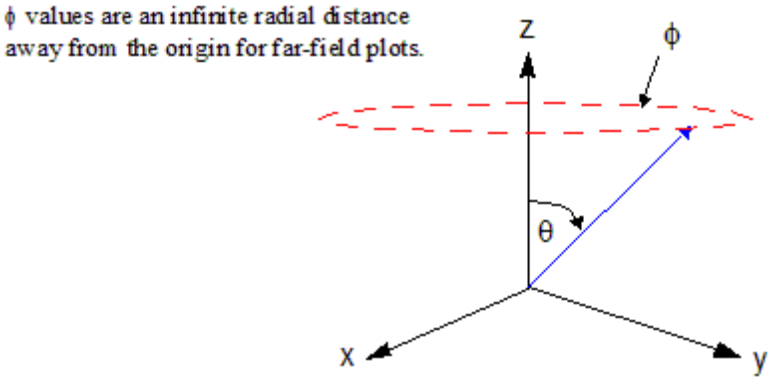
1. Click the ellipsis **[...]** button for the **sweep variable corresponding to theta.**

   To the right, all of the possible values for the theta variable are listed. The values are the result of the range of theta you specified during the infinite sphere's setup.

2. Select the fixed value that theta should take in the plot.

   HFSS will display values for the horizontal cross-section at selected theta cuts of the problem region at a set of phi rotations.

The figure shown below demonstrates the orientation of the sphere on which the field is computed when θ is the fixed variable:



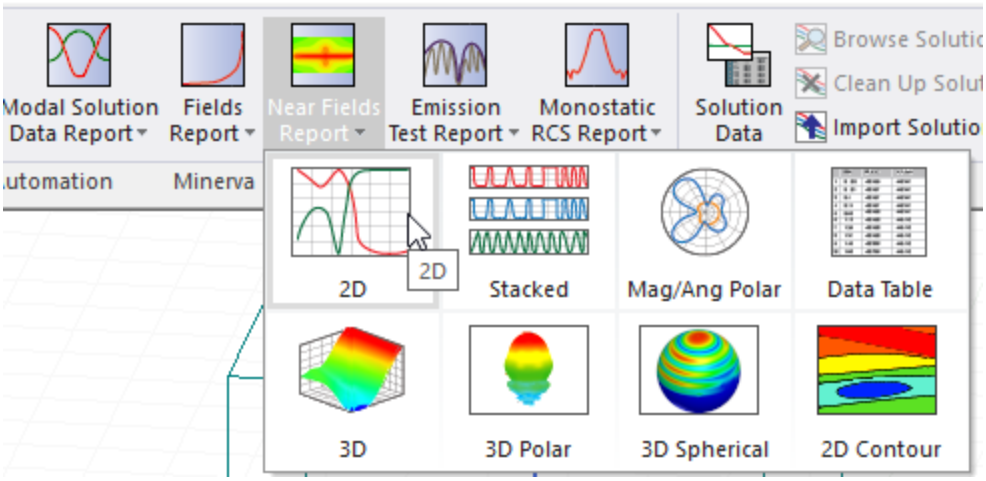
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a Near-Field Quantity to Plot

When plotting near-field quantities, the quantity can be a value that was calculated by HFSS, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.

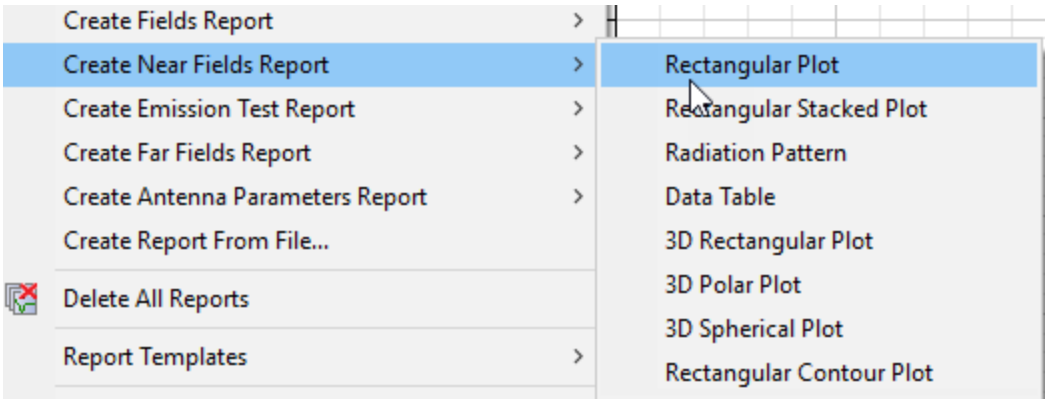
Prerequisites

For the **Near Field Report** icons to appear on the ribbon, or for the **Create Near Field Reports** command to appear on the **Results** menus, the design must contain radiation boundaries, PMLs or Hybrid regions, and must contain a Near Field Setup for sphere, line or rectangle.



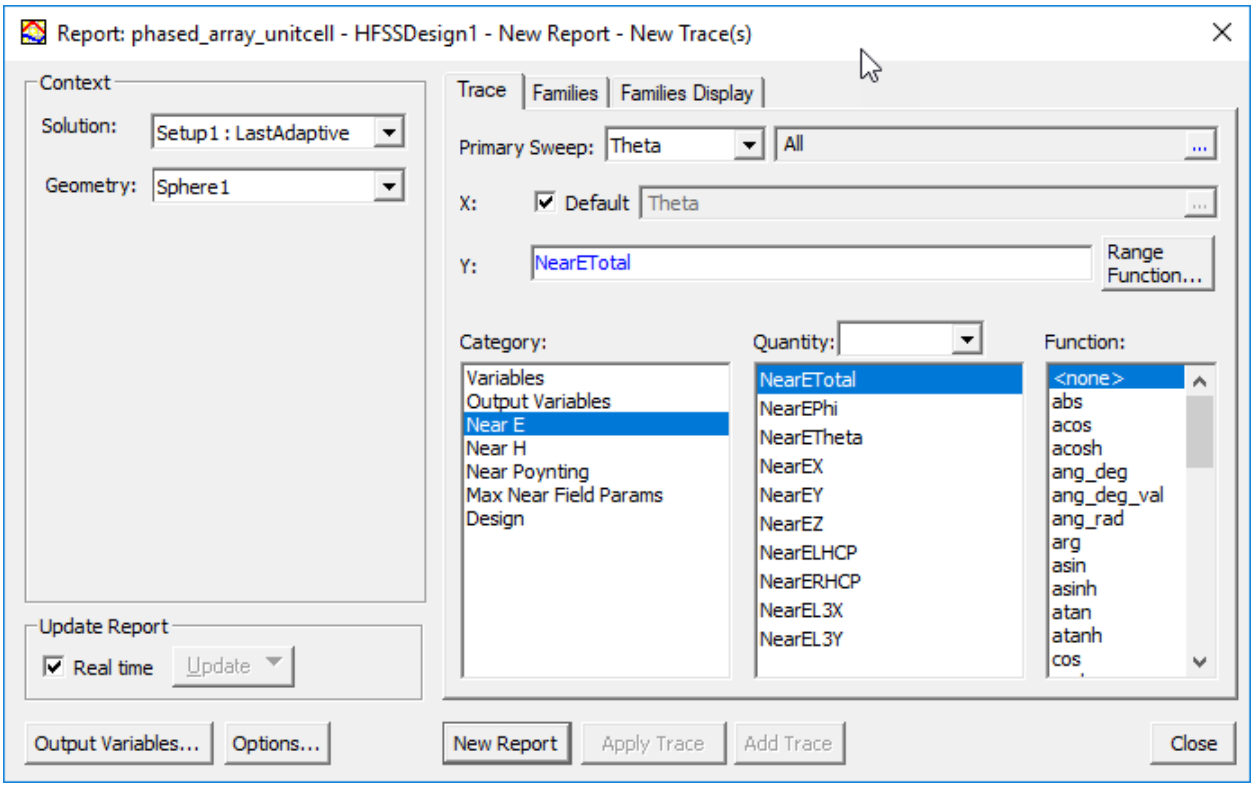
To select a near-field quantity to plot:

1. When you create the report, use the **Results** tab of the ribbon and select the **Near Fields Report** icon to select from the available report types, or if you use the **Results** menus, select **Create Near Fields Report** and select the plot type from the menu.



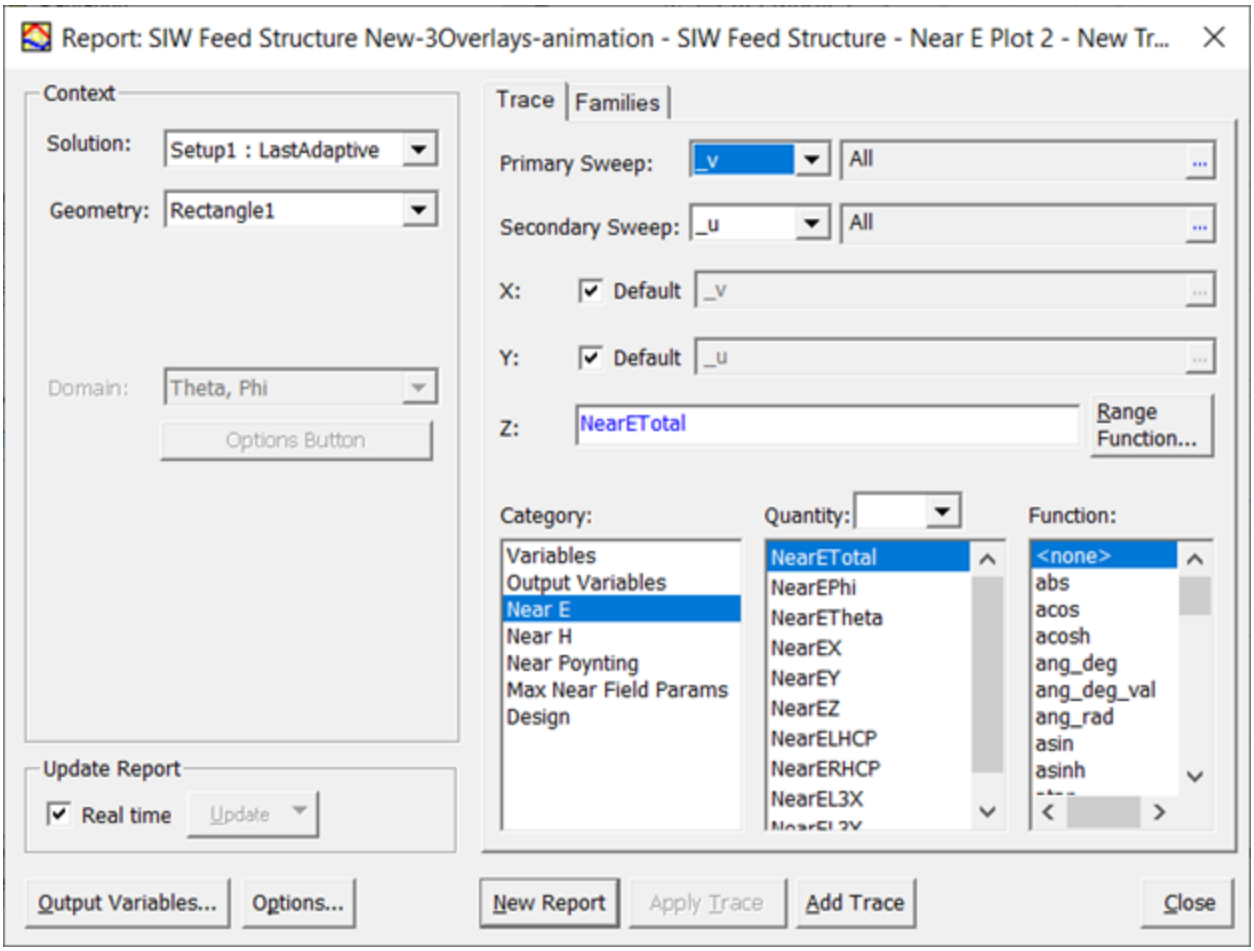
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Report** dialog appears.



When you create a Near Field Rectangular Contour plot, if the Context Geometry is a Rectangle radiation setup and the Primary and Secondary sweep is the combination of _u and _v, then the plot creation dialog is as follows.

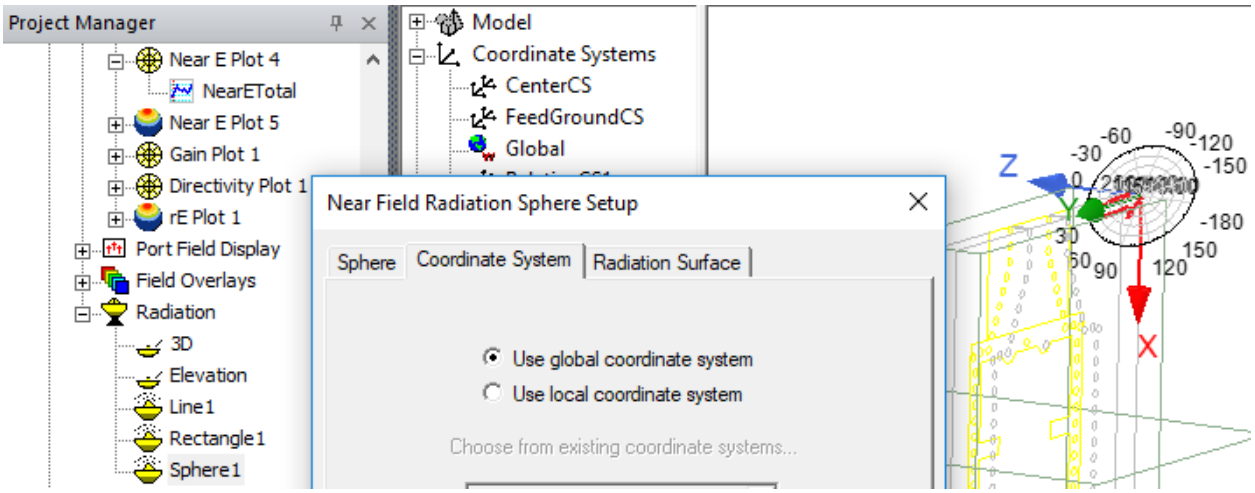
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



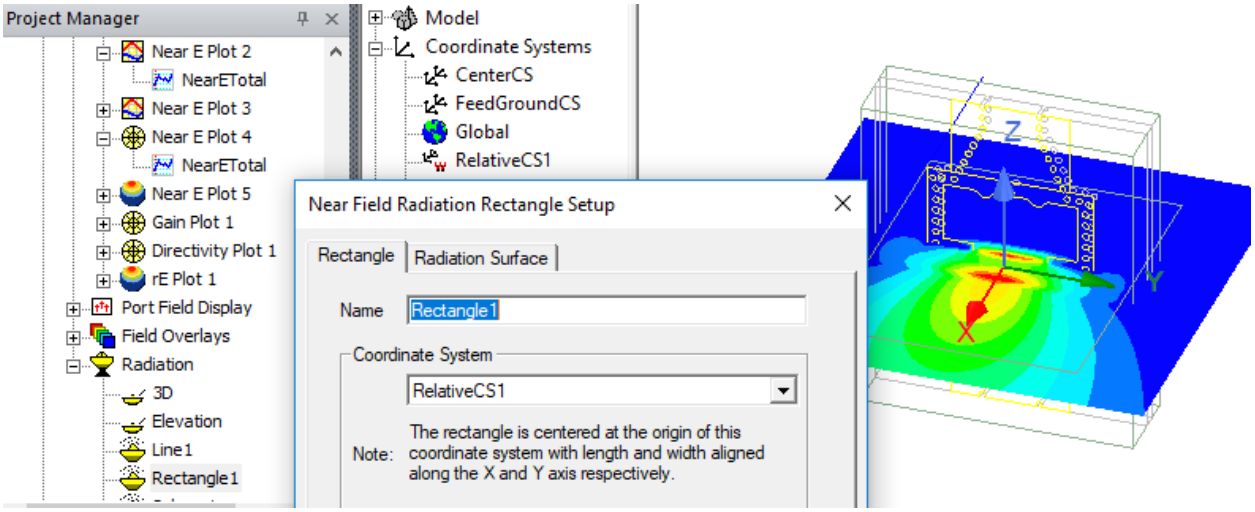
This kind of near field rectangular contour plot can be overlaid in its actual defined rectangle in 3D space in the Modeler window.

2. In the **Geometry** portion, select a predefined Sphere, Rectangle or Line. The geometry selected affects where the plot data is selected and if, where and how the plot will be overlaid on the model, if you choose to do so, after the plot is created. In the following figure, the Near E Polar plot overlaid on the model uses Sphere1 as the Geometry and that geometry uses the Global CS.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

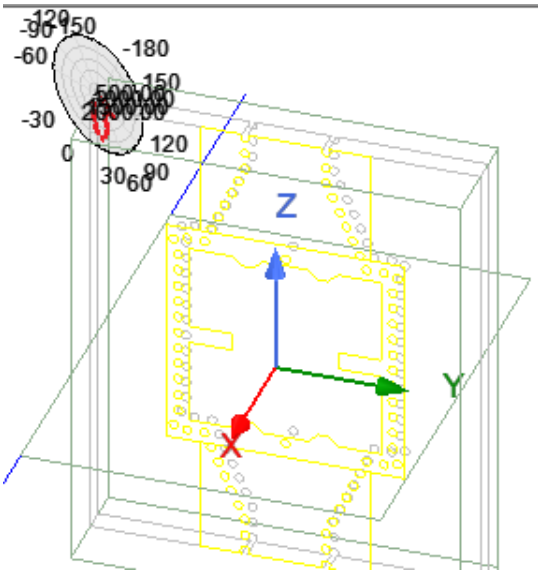


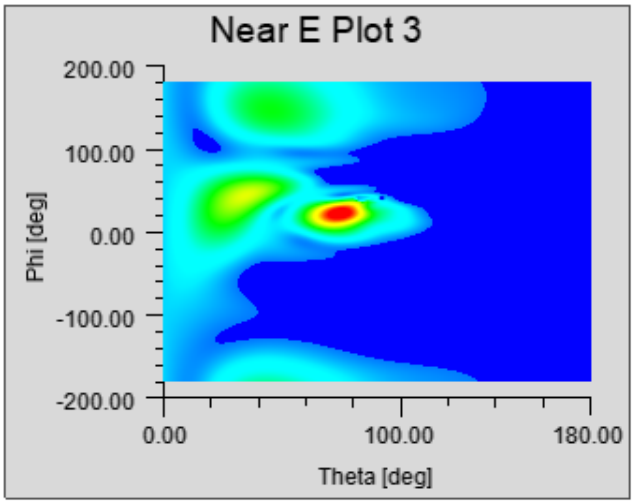
In the following case, Near E Plot 2 Uses Rectangle1, which uses RelativeCS1.



In the case of a plot generated on a line, a plot can be created but not overlaid.
In the case of a 2D Contour Near Field Plot generated on a Sphere, the plot can be overlaid as a plot in the upper right of the model 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. In the **Report** dialog box, select one of the following categories. The selected category provides the default name of the report.

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.
Near E	The radiated electric field in the near field region.
Near H	The radiated H field in the near region
Near Poynting	The Poynting field in the near region. This is 0.5 * E *conj(H).
Max Near Field Params	The maximum radiated electric field in the near region.
Near Normalized Antenna	The resultant plot is: field quantity / (maximum field quantity value over the entire infinite sphere).

4. If you selected the **Near E** category, specify the polarization of the electric field by selecting one of the following types of quantities from the **Quantity** list. The corresponding options appear for Near H and Near Poynting. If you select Max Near Field Params, there are additional **Quantities** for E, H and Poynting.

NearETotal	The combined magnitude of the electric field components.
NearEPhi	The phi component of the electric field.
NearETheta	The theta component of the electric field.
NearEX	The x-component of the electric field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NearEY	The y-component of the electric field.
NearEZ	The z-component of the electric field.
NearELHCP	The dominant component for a left-hand, circularly polarized electric field.
NearERHCP	The dominant component for a right-hand, circularly polarized electric field.
NearECircularLHCP	The polarization ratio for a predominantly left-hand, circularly polarized antenna.
NearECircularRHCP	The polarization ratio for a predominantly right-hand, circularly polarized antenna.
NearEL3X	The dominant component for an x-polarized aperture using Ludwig's third definition of cross polarization.
NearEL3Y	The dominant component for a y-polarized aperture using Ludwig's third definition of cross polarization.

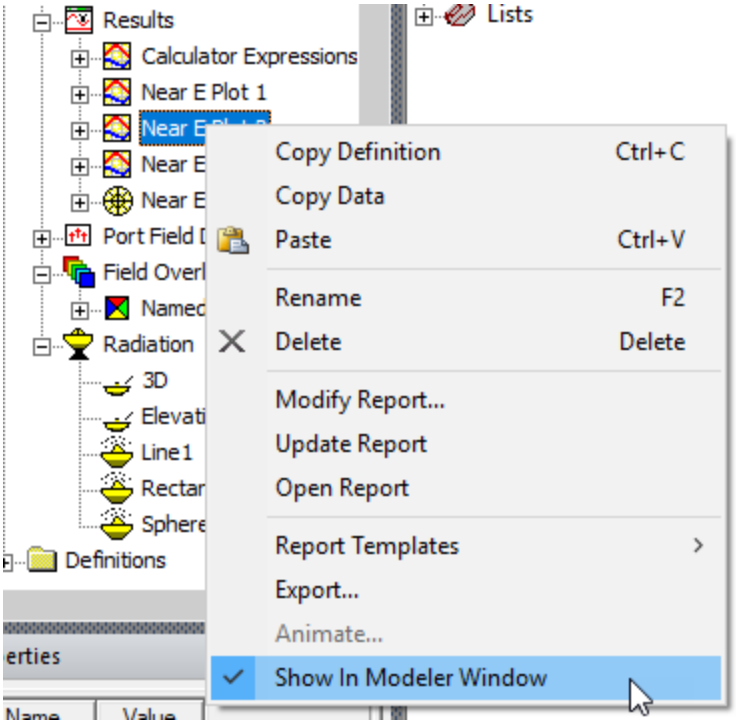
If a Near-field plot takes a long time to plot, be sure to perform **File>Save** when the plot is displayed. This saves the calculated data and permits fast display on subsequent viewings of the plot.

Reports appear under Results, and can be opened as reports. 3D Reports can be overlaid on the model, based on the selected geometry. 2D Near Field Reports on Rectangles can be overlaid as 2D plots.

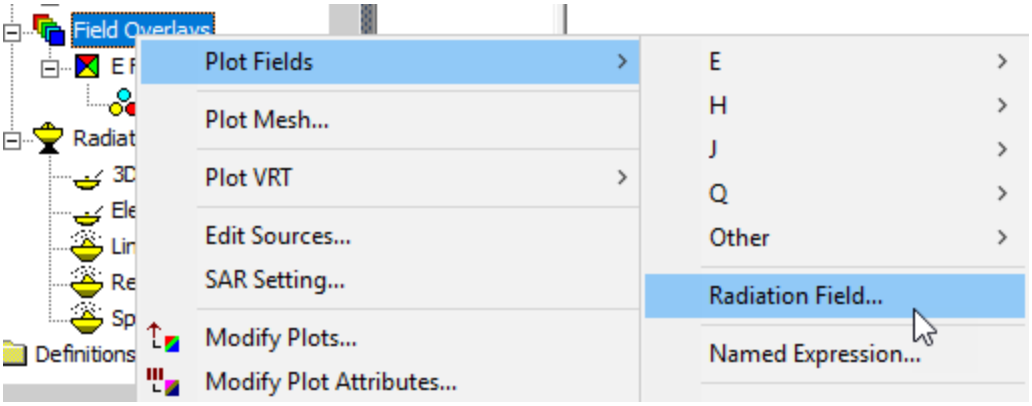
Overlaying Near Field Reports

Depending on the geometry selection and type of plot, some near field plots be shown in the model window. If a plot is appropriate for this purpose, the **Show in Modeler Window** option appears on the **Project** window 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.

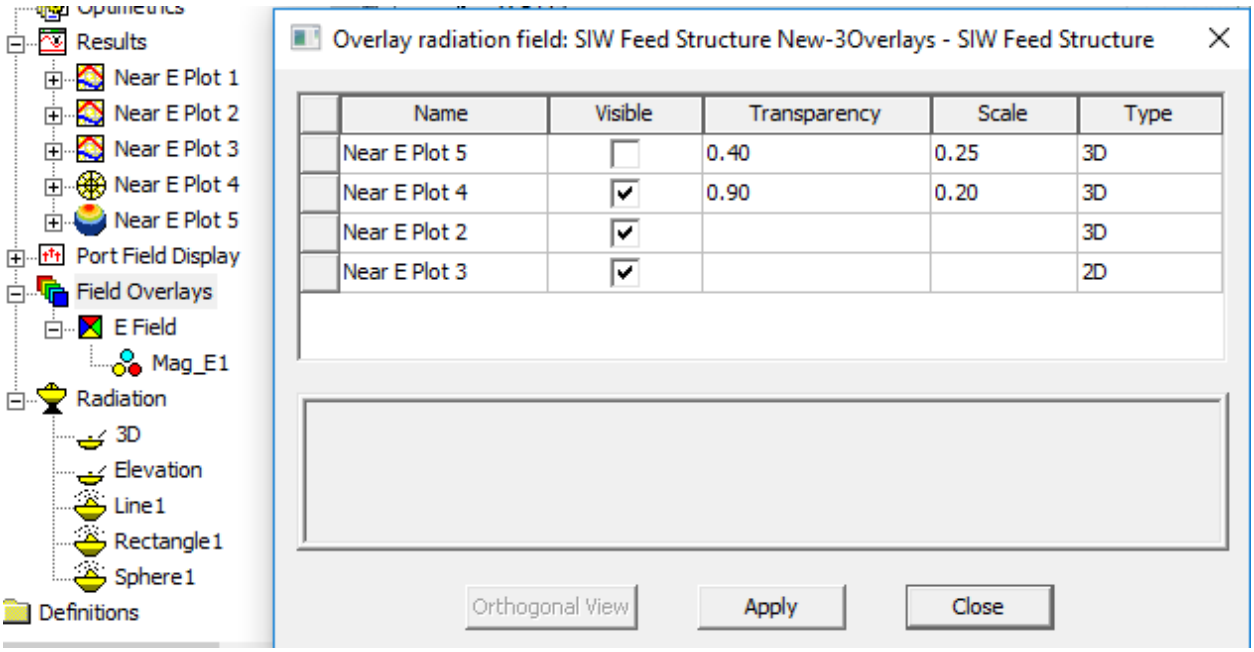


You can also see these plots listed by right clicking on **Field Overlays** in the **Project** tree and selecting **Plot Fields>Radiation Field**.

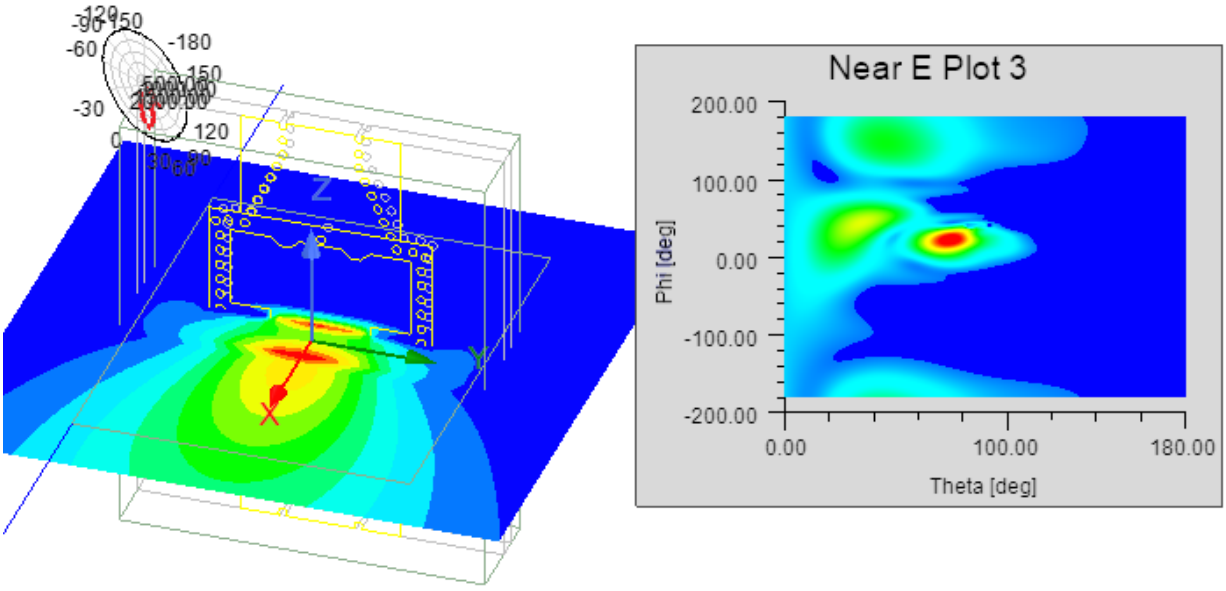


This displays the **Overlay radiation field** dialog box. Notice that it lists **Results** plots that can be overlaid, and not those listed under **Field Overlays**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can display multiple kinds of Plot overlays and Field Overlays at the same time.



Selecting Antenna Parameters to Plot

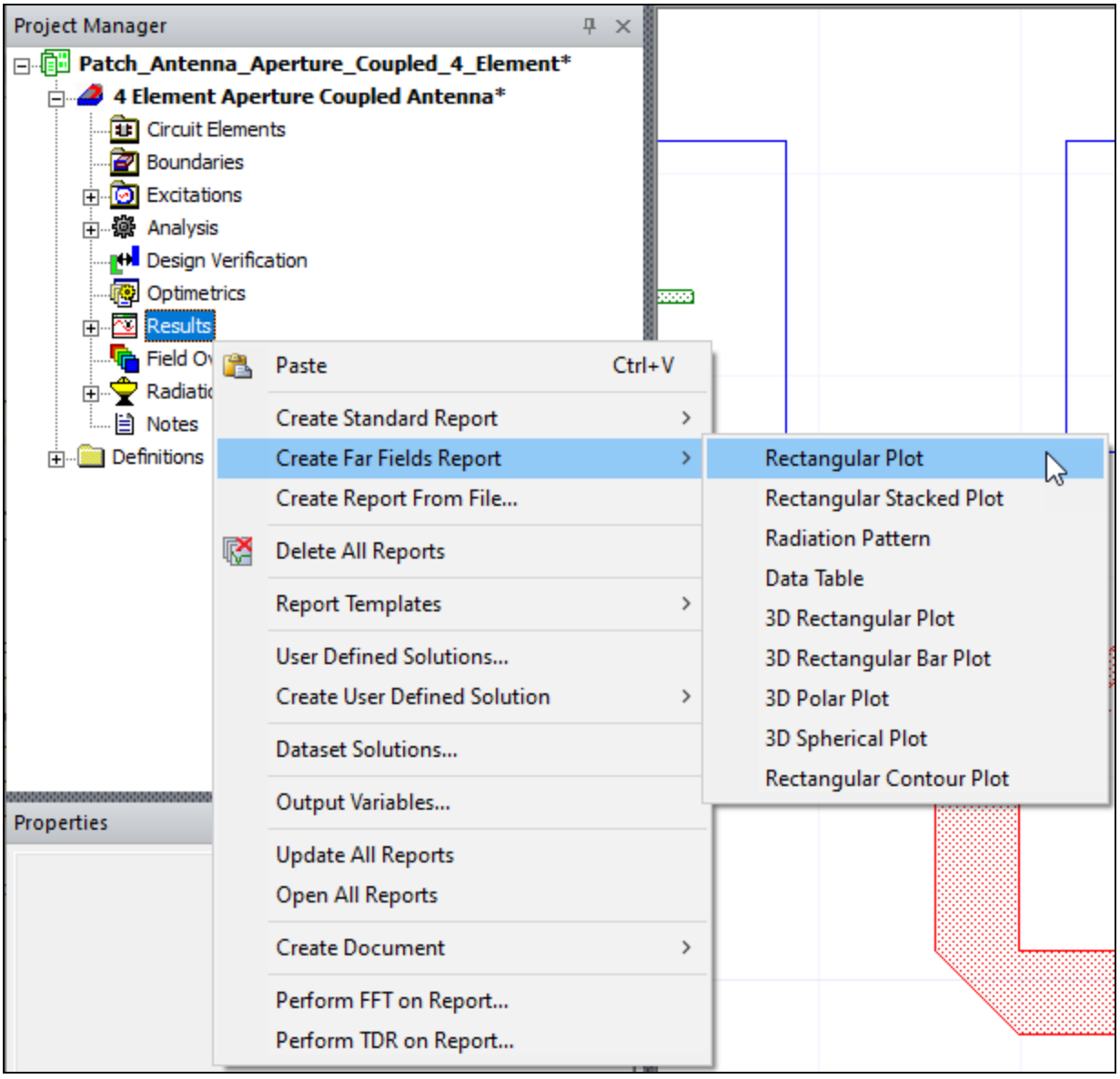
Complete these steps to select and plot antenna parameters.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

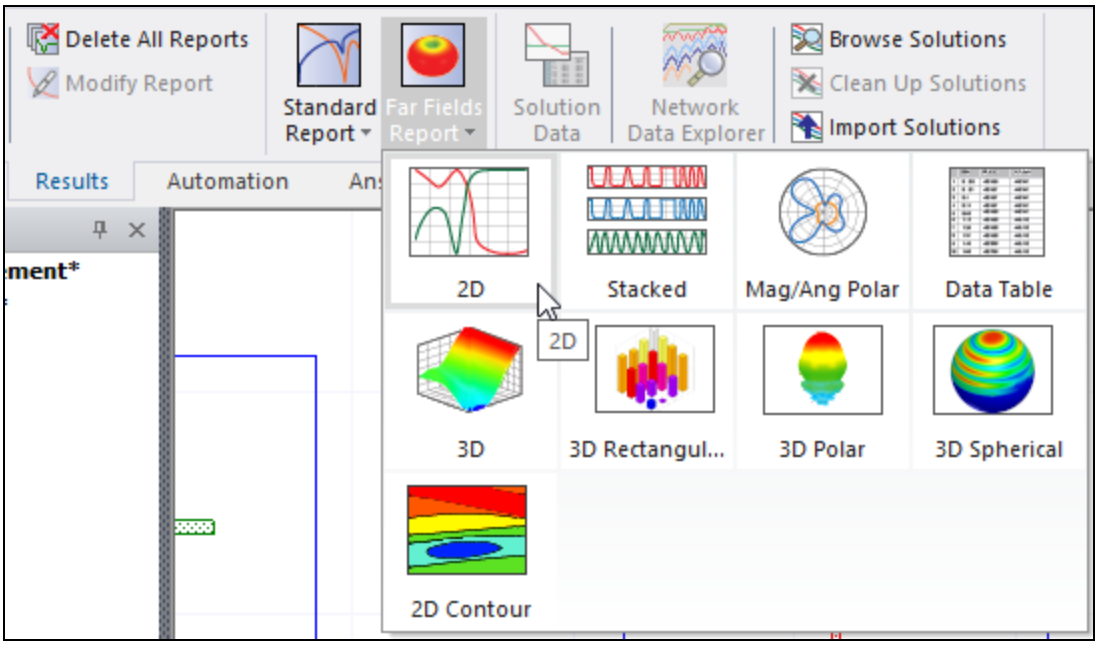
If Far Fields Report options are inaccessible using either method described in step 1, the user will first need to create a coordinate system by adding an **Infinite Sphere** overlay to the design. From the **Project Manager** window, right-click **Radiation** and select **Insert Far Field Setup** > **Infinite Sphere**.

1. Open the **Report** window by doing either of the following:
   - From the **Project Manager** window, expand the **Project Tree** and [***active design folder***] (i.e., **4 Element Aperture Coupled Antenna**). Then right-click **Results** and select **Create Far Fields Report** > [***report type***] (i.e., **Rectangular Report**).



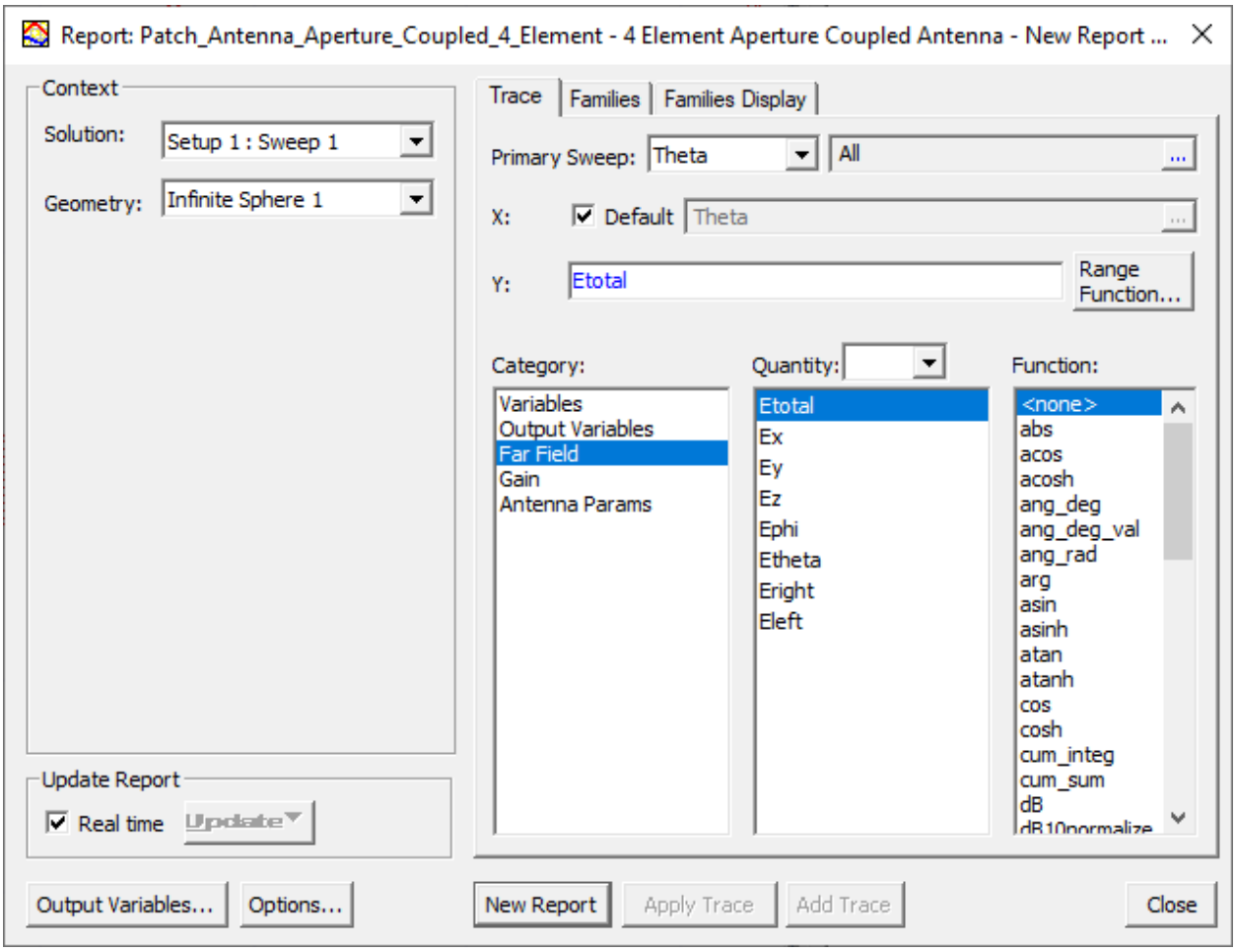
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the **Results** tab, select **Far Fields Report** > [***report type***] (i.e., **2D**).



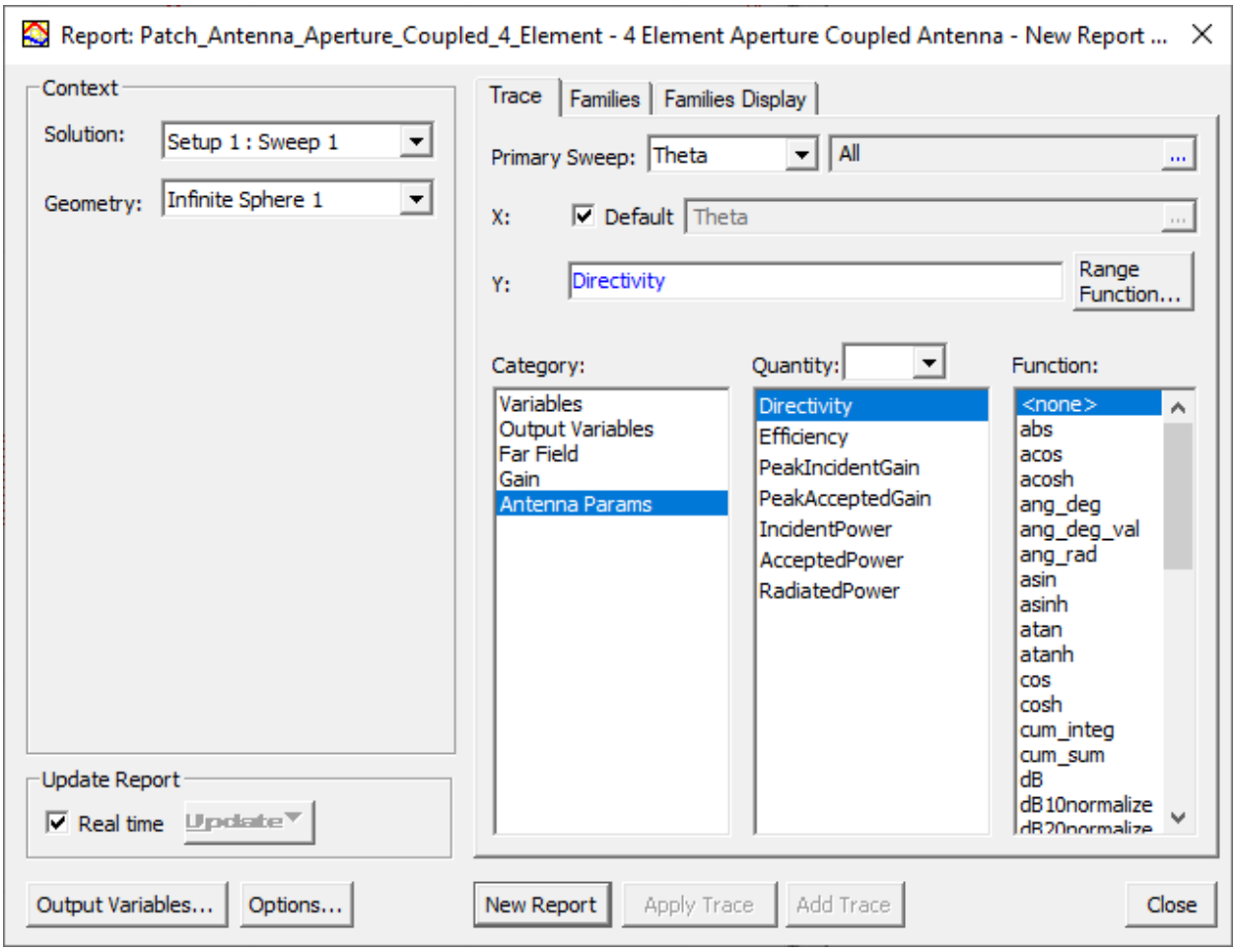
2. From the **Report** window, select **Antenna Params** from the **Category** list to populate the **Quantity** list with selectable antenna parameters.

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.

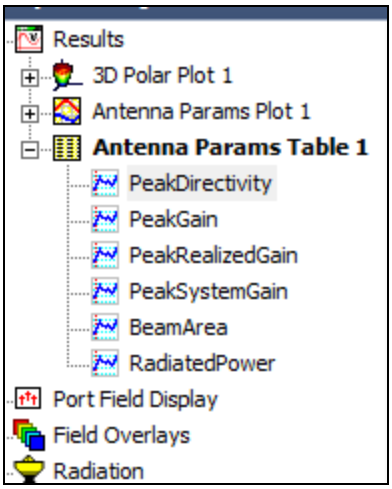
3. Select one or more parameters from the **Quantity** list.



4. If appropriate, select one or more options from the **Function** list.
5. Make any other selections, as necessary.
6. Click **New Report**.
7. If necessary, return to the **Report** window to select additional options from the **Quantity** and **Function** lists. Then click **Add Trace**. Each additional trace appears in the report and

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**.



Antenna Parameters

HFSS can calculate the following antenna properties:

- Peak directivity
- Peak gain
- Peak realized gain
- Radiated power
- Accepted power
- Incident power
- Radiation efficiency
- Total efficiency
- System efficiency

Selecting an Emission Test Quantity to Plot

1. When you create the report, select Emission Test.
2. In the **Report** dialog box, select one of the following categories and apply an appropriate Quantity. The selected category provides the default name for the report.

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.
Sphere	A sphere of 1, 3, 10, or 30 meters, or of the same dimensions and RBS Simple or RBS exact (where RBS is random binary [bit] sequence).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylinder A cylinder of 3 or 10 meters, or of the same dimensions and PRBS Simple or PRBS exact.

3. Select a **Function** for the quantity from the function list.
4. For Emission Test, the **Report** dialog also contains a button for specifying the digital signal options. The default values are a rise time of 0 seconds, and a hold time of 1 second. To specify other values, click **Digital Signal...**.

This displays the **Digital Signal Options** dialog box. It contains fields for the rise time and hold time.

5. **OK** the specified values or Cancel, Use Defaults, or Save As Default as appropriate.

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:

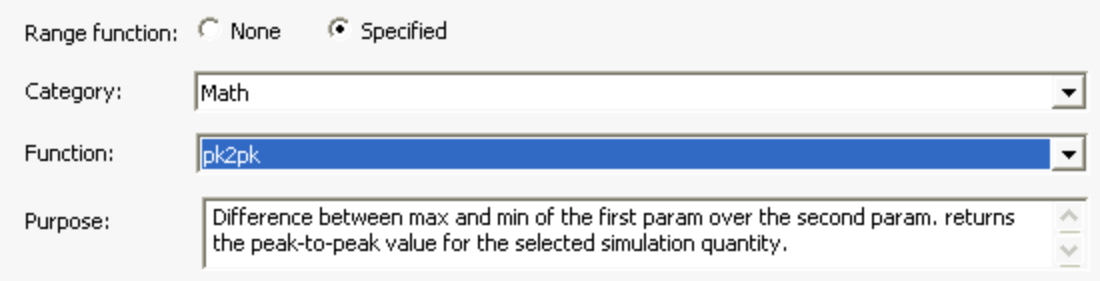
1. Click **Range Function** in the *Report* dialog box.

   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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Given a selected Function, and Category, the **Set Range Function** dialog displays a text field that explains the Purpose of the function. See figure above.

Selecting a function causes the display of a description in the **Purpose** field. If the function requires a value (such as the XatYVal Math function or the pw_minus_max Pulse Width function), the following table the function field displays the name, editable value field, unit, and description.

4. Use the **Over Sweep** drop-down menu to select from available sweeps.
5. To select from available Sweeps, or to edit them, use the ellipsis [**...**] button and uncheck **Use All Sweeps**.

   This enables a list of the sweeps. The sweep(s) you select is displayed on the Over Sweep line. You can use the buttons to **Clear All Selections** or **Select All** sweeps.

6. Select the Sweeps **Default** or **Edited** radio buttons to specify whether to accept the default or edited sweeps.
7. To edit the sweeps further, select the ellipsis button to display an *Edit Sweep* dialog box.

   For frequency variables, this lets you specify a single value, linear step, linear count, decade count, octave count, or exponential count. You can **Add** legal values to the list of sweep values, **Update** the list for changes, or **Delete** selected entries.

8. Click **OK** to apply the range function.

Range Functions

The following table shows the **Functions** according to their **Categories**. The most commonly used range categories are **Math** and **Radiation**. Other functions could be used if needed. Use the category links to navigate to tables with definitions of functions.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Period	per, pmax, pmin, prms
AC	gainmargin, phasemargin, gaincrossover, phasecrossover, lowercutoff, uppercutoff, bandwidth, peakgain, peakgainfreq.
Radiation	xdb10bandwidth, xdb20bandwidth, lSidelobeX, lSidelobeY, rSidelobeX, rSidelobeY

Math Functions	
***avg**	Returns the average of the values of the selected quantity. avg = (Area between the curve and the X-axis) / (X length of the curve)
avgabs	Returns the mean of the absolute value of the selected quantity.
integabs	Absolute value of integral.
max	Returns maximum value of the simulation quantity.
mean	Returns the average in the set of quantities selected. mean = sum(all y-value) / (number of y-values)
min	Returns the minimum value of the simulation quantity.
rms	Returns the root mean square value of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
ripple	Returns the ripple factor (AC RMS/Mean) for the selected quantity.
pkavg	Returns the ratio 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.
YAtXMin	Returns the Y value at minimum value of X.
YAtXVal	Returns the Y value at the first occurrence of X value.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiation Functions	
lsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
lsidelobeY	The 'y' value for the left side lobe: the next highest value to the left of the max value.
rSidelobeX	Returns the X value of right side-lobe occurrence.
rSidelobeY	Returns the Y value of right side-lobe occurrence.
xdb10beamdwidth	Width between left and right occurrences of values 'x' db10 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
xdb20beamwidth	Width between left and right occurrences of values 'x' db20 from max. Takes 'x' as argument (3.0 default) To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.

Pulse Width Functions

> **Note:**
>
> In this table, the functions with the asterisk (*) do not appear on the Range Function drop-down menu. They can still be used via text entry.

pulsefall9010	Returns the pulse fall time of the selected quantity according to the 90%-10% estimate.
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%.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

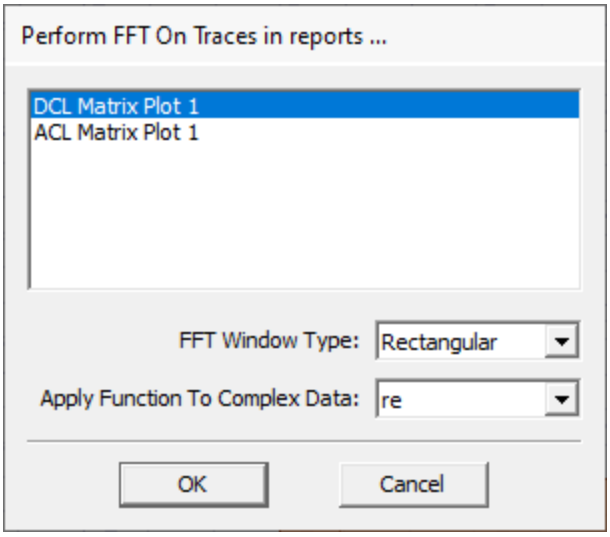
The Eye Measurement category of range functions provide the means to evaluate several characteristics of eye diagrams. Each of the Eye Measurement functions includes the following parameters. Specify the Value by editing the Value text field for the parameter name. Specify the unit for the parameter 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.

Perform FFT on a Report

You can perform FFT on an existing 2D plot by using the **Results** > **Perform FFT** command. You can perform TDR on an existing 3D plot by using **HFSS 3D Layout** > **Results** > **Perform FFT**. This opens the **Perform FFT on Traces in Reports** 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.

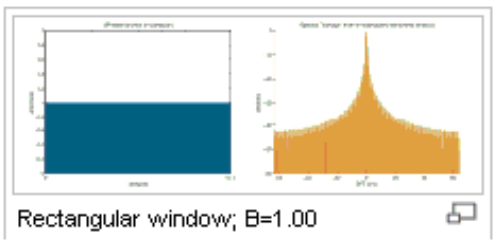


1. Select the report you want from the list.
2. Select the FFT window type to apply. Winding functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies.
3. Select the function to apply to complex data.

   The new report appears under **Results** in the **Project Manager**. The new report name prefixes FFT to the name of the original report. Trace names are also prefixed with FFT.

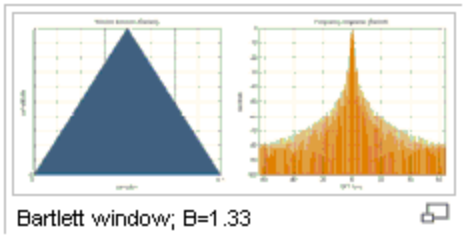
FFT Window Functions

The window type list for **Perform FFT on Report** are listed below.

Window Function	Preferred Use
Rectangular	A low dynamic range function offering good resolution for signals of comparable strength. Poor when signals have very different amplitudes. *w*(*n*)=1 
Tri	A Bartlett window with the endpoints valued at zero.

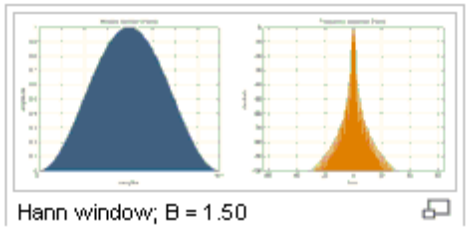
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use



Van Hann — A moderate dynamic range function, designed for narrow band applications.

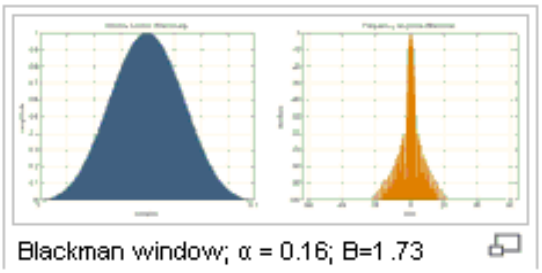
$$w(n) = 0.5\left(1 - \cos\frac{2\pi n}{N-1}\right)$$



Blackman — A high dynamic range function, with lower resolution, designed for wide band applications.

$$w(n) = \alpha_0 - \alpha_1 \cos\left(\frac{2\pi n}{N-1}\right) + \alpha_2 \cos\left(\frac{4\pi n}{N-1}\right)$$

where a_0 =(1-α)/2; α_1=1/2; α_2=α/2

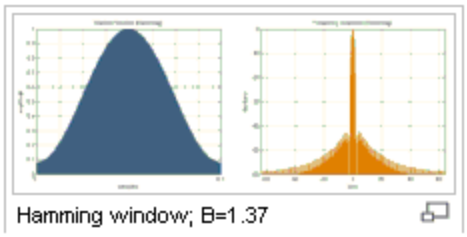


Hamming — A moderate dynamic range function, designed for narrow band applications. It minimizes the maximum sidelobe.

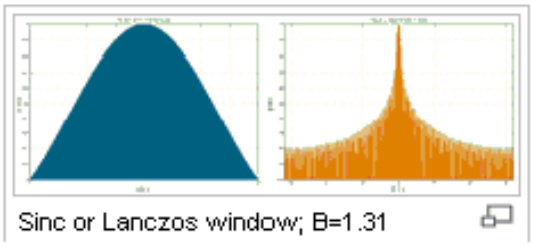
$$w(n) = 0.54 - 0.46\cos\left(\frac{2\pi n}{N-1}\right)$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use



Lanczos — The Lanczos window offers a windowed form of the infinite sinc filter, providing the central lobe of a horizontally stretched sinc, sinc(x/a) for -a £ x £ a.



Weber — Decomposes functions into Laguerre series (orthogonal polynomials), and then uses FFT to evaluate the transform; for more details, see H. Weber, "Numerical Computation of the Fourier Transform Using Laguerre Functions and the Fast Fourier Transform", Numer. Math., Vol. 36, pp. 197--209 (1981).

Welch — This approach applies a parabola-shaped window to the frequency domain data. It is based on the Bartlett method but splits the signal into overlapping segments, which are then windowed. The intent is to balance the influence of data in the center of the function.

Apply FFT to Report Functions

The functions that can be used in the Perform FFT on Traces in Reports dialog box are listed below.

ang_deg	Angle (phase) of a complex number, cut at +/-180
ang_deg_val	Angle (phase of a complex number in unitless degree values. Returns simple numbers.
ang_rad	Angle in radians
arg	

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
cang_deg_val	Cumulative angle (phase) of the first parameter of the selected simulation quantity in unitless degree values. Returns simple numbers.
cang_rad	Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable) Returns a double precision value.
dB(x)	20*log10(|x|)
dB 10normalize	10*log [normalize(mag(x))]
dB 20normalize	20*log [normalize(mag(x))]
dBc	
GetGroupDelay	
im	Imaginary part of the complex number
mag	Magnitude of the complex number
normalize	Divides each value within a trace by the maximum value of the trace. ex. normalize(mag(x))
re	Real part of the complex number

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. If you want to plot the polar angle of a complex simulation result, S_{11} 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(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. 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.

Perform TDR on Report

You can perform TDR on an existing 2D plot by using the **Results** > **Perform TDR on Report** command. You can perform TDR on an existing 3D plot by using the **HFSS 3D Layout** > **Results** > **Perform TDR on Report** command. This opens a **Perform TDR on Traces in reports** dialog box.

Note:

The accuracy of the early time TDR results depends on the highest frequencies simulated in the frequency sweep. Use the **Time Domain Calculation** dialog box to set up the frequency sweep. Set the maximum simulation frequency for your desired rise time.

1. Select the report you want from the list in the dialog box.
2. Specify the input signal as **Step** or **Impulse**.
3. Set the **Rise Time**.
4. In the **Type** drop-down menu, select which window type to apply.

   Windowing functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies.

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 specify a window width, enter a percentage into the **Width (%)** field.
6. If you select the Kaiser function, specify a number in the **Kaiser Parameter**.

The new report displays and appears in the Project Manager tree. The new report name prefixes TDR to the name of the original report. Trace names are also prefixed with TDR.

Window Functions and Time Domain Plotting

HFSS and SIwave allow for time-domain plotting of S-parameters. Often, this feature is used to calculate a step response or time-domain reflectometry (TDR) plot of the structure being simulated. Fourier analysis provides the mathematical mechanism for transforming frequency sweep data to a time-domain plot, but two approximations are involved. First, the transform is between two sets of discrete data points, as opposed to continuous waveforms. Second, the frequency sweep data cannot have infinite bandwidth, but must truncate at some upper limit. This section discusses the implications of these approximations, and provide information for successful time-domain plotting.

Note: Frequency sweep data consists only of positive frequencies, but the negative frequencies are simply the complex conjugate of the positive: $S(-f) = S^*(f)$

This is true for any frequency-domain function when the corresponding time-domain waveform is real-valued.

Transforming Frequency- To Time-Domain

It is easier to make generalizations about the effect of finite bandwidth if we have continuous functions. Consequently, we will initially assume our frequency- and time- domain data is continuous, and defer discussion of the effects of discretization until later. With a continuous-time sweep over an infinite bandwidth, we could – at least in principle – calculate a time-domain response by multiplying our sweep data $S(f)$ with the spectrum of a time-domain excitation function and evaluating the inverse Fourier integral:

$$f(t) = \int_{-\infty}^{\infty} S(f)E(f)e^{2\pi jft}df$$

In practice, however, sweep data does not extend to infinite frequencies and is restricted to a bandwidth b. If we simply assume that the spectrum is zero-valued outside of the bandwidth, we can interpret the data as an infinite sweep that has been multiplied by a rectangular “window” function $W(f)$, with a value of 1 within the bandwidth and a value of 0 otherwise.

$$f(t) = \int_{-\infty}^{\infty} W(f)S(f)E(f)e^{2\pi jft}df = \int_{-b}^{b} S(f)E(f)e^{2\pi jft}df$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This process is illustrated in Fig. 1, assuming that $S(f)$ $E(f)$ corresponds to an ideal unit step function in the time domain. In Fig 1.a, the frequency spectrum is truncated beyond a certain upper limit. Since multiplication in the frequency domain corresponds to convolution in the time domain, this has the effect of convolving the time-domain step with a sinc function – the inverse Fourier transform of the rectangle (Fig 1.b). The final result is an edge with a finite rise time and some oscillation.

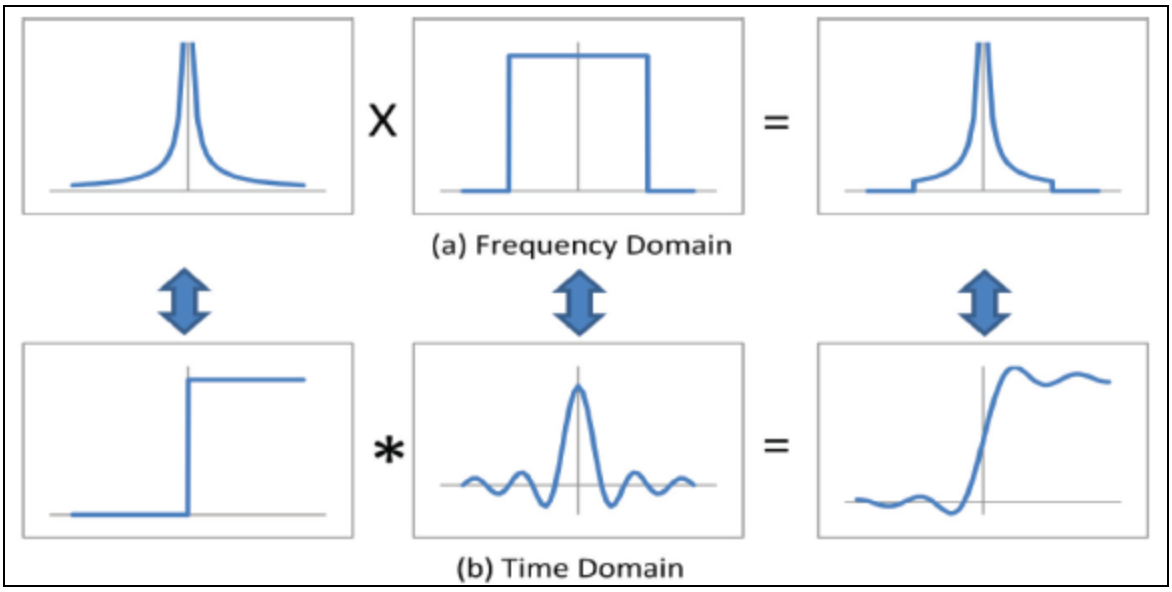
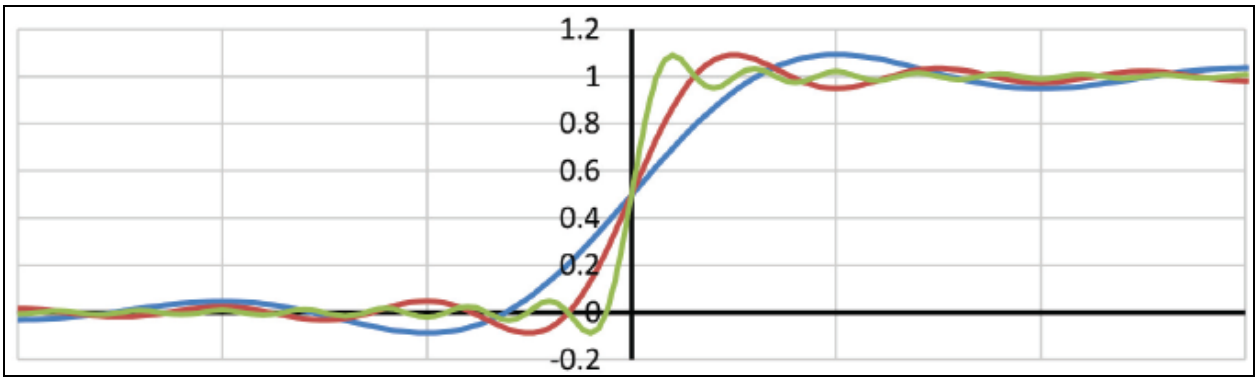


Figure 1. Multiplying the spectrum of a step function with a rectangular window produces a finite edge in the time domain

If the sweep is extended to higher frequencies – making the window function wider – the corresponding sinc pulse more closely approaches an impulse, and the time-domain edge becomes sharper. However, the oscillation never disappears for any finite sweep. Fig. 2 shows a step response for increasingly wider bandwidths.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2. Increasing the width of the rectangular window makes the time-domain edge sharper, but does not eliminate the oscillation

Some distortion of the true time-domain waveform is unavoidable if the frequency sweep does not include the entire bandwidth of the signal, but there are other window functions besides the rectangle which distort the time waveform in ways which may be more desirable. In particular, it would be nice to reduce the spurious oscillation. The next sections will describe the window functions available and discuss their effects.

Window Functions

The window functions available are plotted in Figs. 3 and 4, and their expressions are given in the Appendix. All the window functions have a spectral width *w* and are zero-valued for I*f* I > *w*/2. In addition to truncating the data outside of the bandwidth, the non-rectangular windows filter the spectrum inside. The windows differ from each other in how strongly they attenuate the spectrum as the frequency approaches the upper limit. The Kaiser window has a parameter, which controls how sharply it decays. For $\alpha = 0$, the Kaiser window is equivalent to the rectangular window; for $\alpha = 5.4414$, it is equivalent to the Hamming window; and for $\alpha = 8.885$, the Blackman window.

Although the TDR Options dialog allows for windows that are narrower than the bandwidth of the simulation, it is generally best to set the window width to 100% and take full advantage of the available bandwidth.

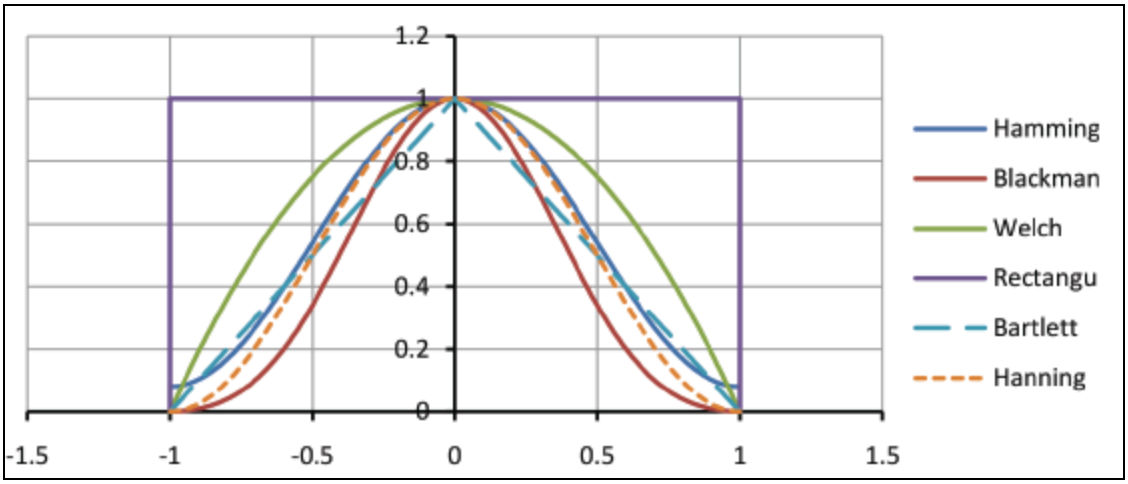


Figure 3. Window functions with a width of *w*=2

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

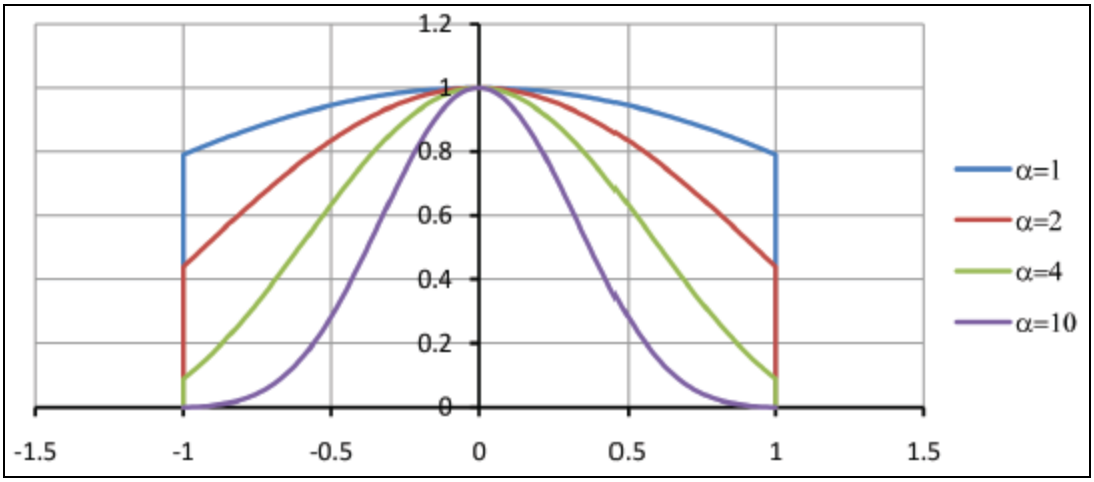


Figure 4. The Kaiser window for a width of *w*=2 and varying values

Because the spectral width *w* includes both positive and negative frequencies, it is twice the bandwidth of the sweep, *b*, which is equal to the (positive) upper frequency limit.

Ideal Step Response

It is immaterial whether we think of the window as multiplying the frequency sweep data, with the spectrum of the time-domain excitation having infinite bandwidth, or if we instead imagine we have infinite sweep data and a windowed excitation spectrum. With the latter interpretation, we can examine the effects of different windows on an ideal step without concern for what the sweep data looks like.

We will apply different windows to an ideal step function, which is approximated in HFSS and SIwave by choosing an edge and setting the rise time to 0. We continue to assume that we have a continuous spectrum, and will defer a discussion of the effects of discretization until later. The effect of the Welch window is shown in Fig. 5.

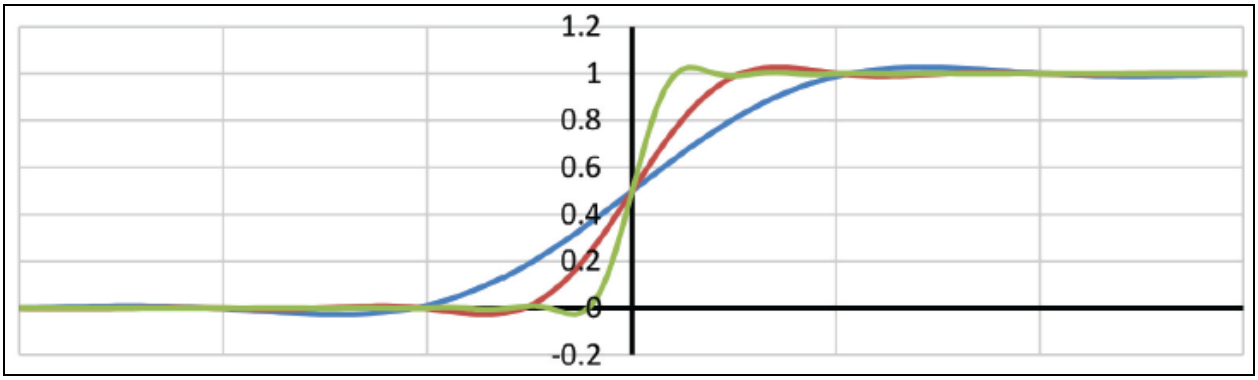


Figure 5. The effect of Welch windows of three different widths on an ideal step

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fig. 5 shows that the Welch window has substantially decreased the signal oscillation that was seen with the rectangular window. As Fig. 6 below demonstrates, the Blackman window results in almost no oscillation.

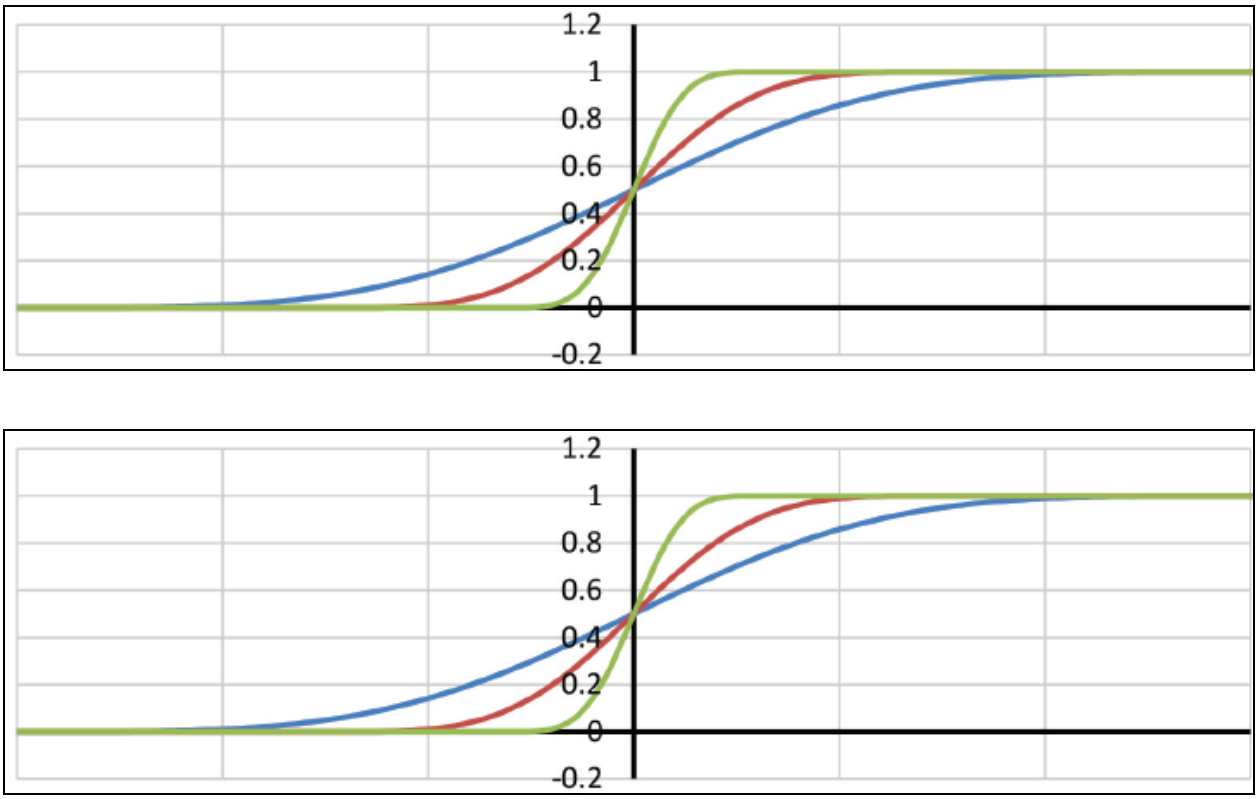
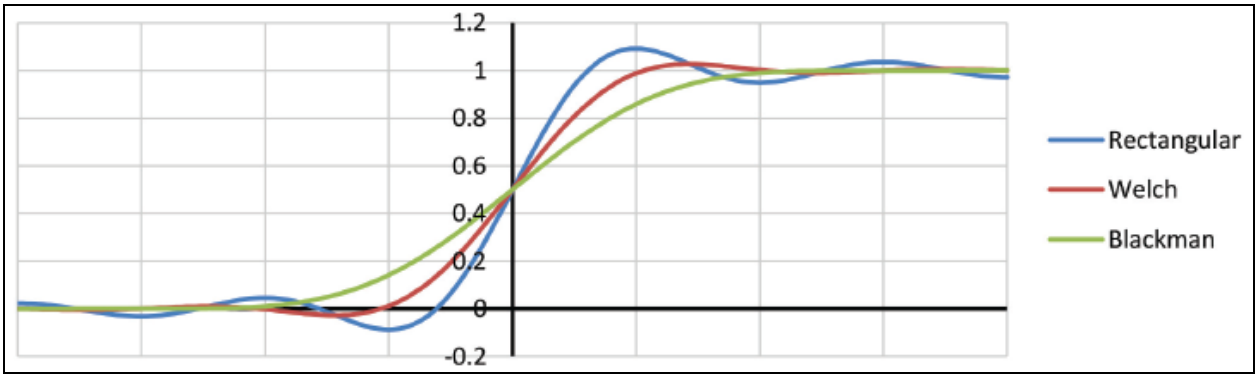


Figure 6. The effect of Blackman windows of three different widths on an ideal step.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

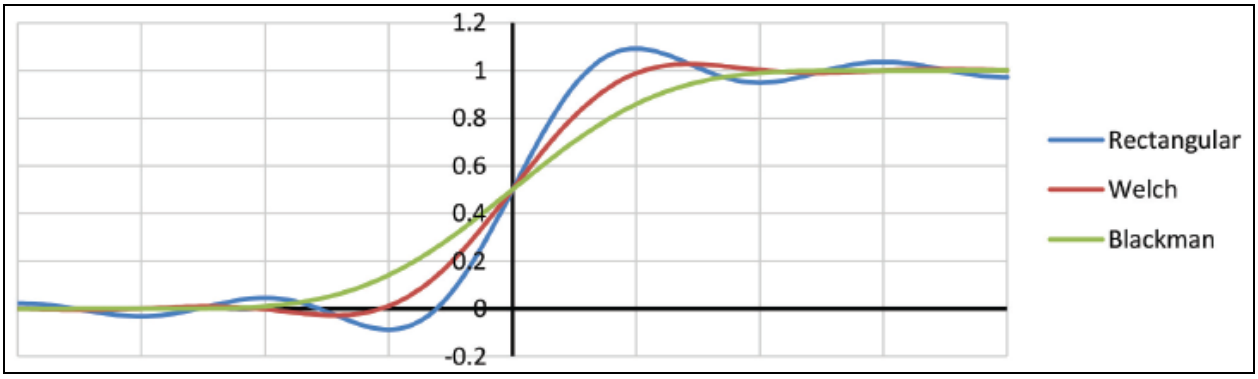


Figure 7. Step response for three different windows, each with the same bandwidth.

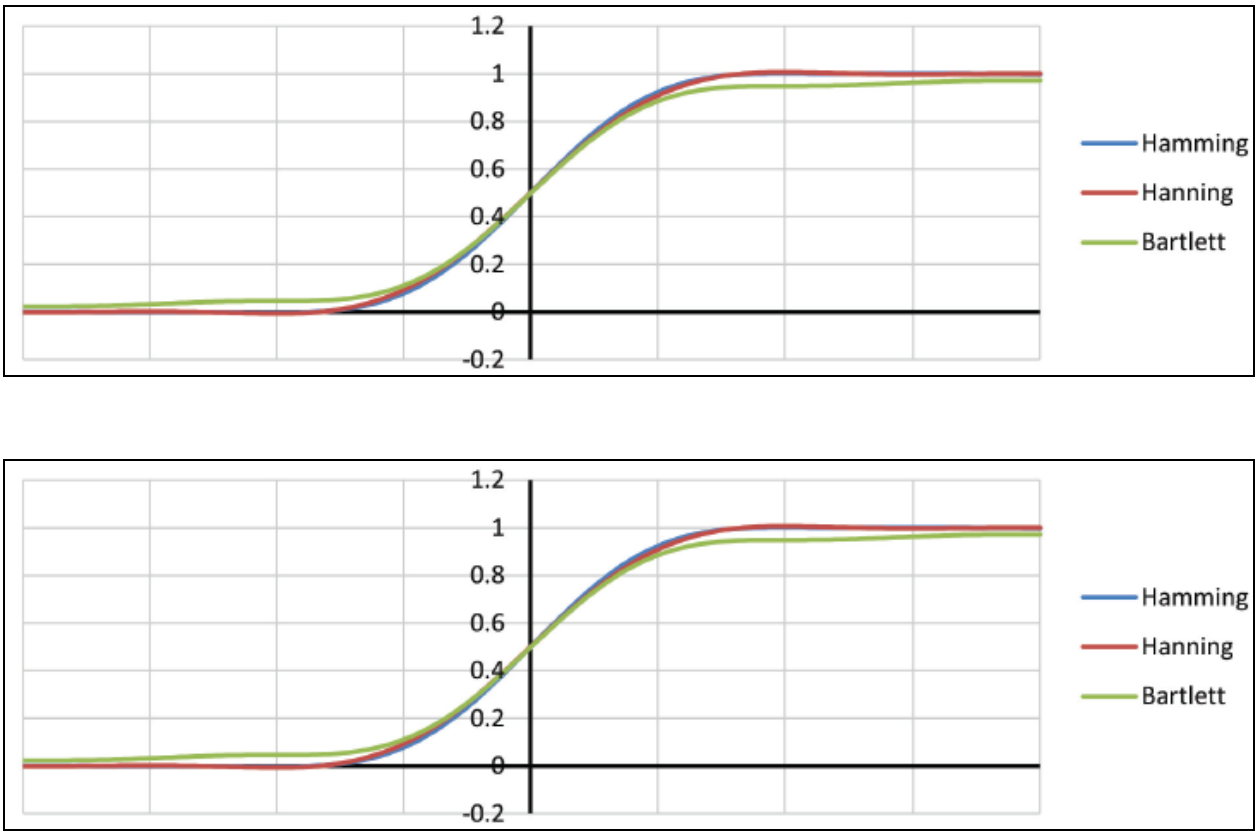


Figure 8. Hamming, Hanning, and Bartlett windows of equal bandwidth

As Fig. 8 suggests, the difference between Hamming and Hanning windows is usually quite small. The Bartlett window is generally not recommended, as it distorts the signal in the vicinity of the edge without providing any advantage over the Hamming and Hanning windows. The Kaiser window gives edges that are slower and less oscillatory with increasing α.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The rectangular, Welch, Hanning, and Blackman windows are sufficient to provide a good sampling of the edge-rate vs. oscillation tradeoff. Table 1 quantifies the characteristics of these windows on an ideal step. With the exception of the Blackman window, it is possible to derive reasonably simple expressions for the step response. In Table 1, *b* is the bandwidth or upper frequency limit of the sweep and Si(χ) refers to the sine integral function:

$$\mathrm{Si}(x) = \int_0^x \frac{\sin z}{z} dz$$

Table 1. Characteristics of selected window functions for continuous time.

Window	*Step Response $f_s(t)$*	*10-90 Edge Rate*	*Max Overshoot*
Rectangular	$\frac{1}{2} + \frac{\mathrm{Si}(2\pi bt)}{\pi}$	$\frac{0.45}{b}$	8.95%
Welch	$\frac{1}{2} + \frac{\cos(2\pi bt)}{2\pi^2 bt} - \frac{\sin(2\pi bt)}{4\pi^3 b^2 t^2} + \frac{\mathrm{Si}(2\pi bt)}{\pi}$	$\frac{0.70}{b}$	2.70%
Hanning	$\frac{2\pi + 2\mathrm{Si}(2\pi bt) - \mathrm{Si}(\pi - 2\pi bt) + \mathrm{Si}(\pi + 2\pi bt)}{4\pi}$	$\frac{0.97}{b}$	0.64%
Blackman	--	$\frac{1.19}{b}$	0.02%

Note that in the expressions for the step response, the time variable is always multiplied by the bandwidth. Changing the bandwidth scales the time response, but does not affect the shape of the edge.

Finite Edge Response

Finite edges can be simulated by providing a nonzero value for the rise time. For finite edges, the same edge rate vs. oscillation tradeoff applies. However, the spectrum of a finite edge declines with frequency at a faster rate than an ideal step. As a result, modest amounts of overshoot can be achieved even with a rectangular window. The continuous time finite edge response of a rectangular window is given by

$$f_e(t) = \frac{1}{2} - \frac{\mathrm{Si}\big(2\pi b(r-t)\big)}{\pi} + \frac{\cos(2\pi bt) - \cos\big(2\pi b(r-t)\big) + 2\pi bt\left(\mathrm{Si}\big(2\pi b(r-t)\big) + \mathrm{Si}(2\pi bt)\right)}{2\pi^2 br}$$

The value of the edge response at $t = 0$ 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.

$$f_e(0) = \frac{1}{2} + \frac{1 - \cos(2\pi br)}{2\pi^2 br} - \frac{\mathrm{Si}(2\pi br)}{\pi}$$

Along with the overshoot, f_e (O) is a useful metric for describing how closely the finite edge response approximates the ideal case, for which f_e (O) = O. The degree to which the windowed edge approximates an ideal finite edge depends only on *br*, the dimensionless product of the bandwidth and the rise time (Fig. 9).

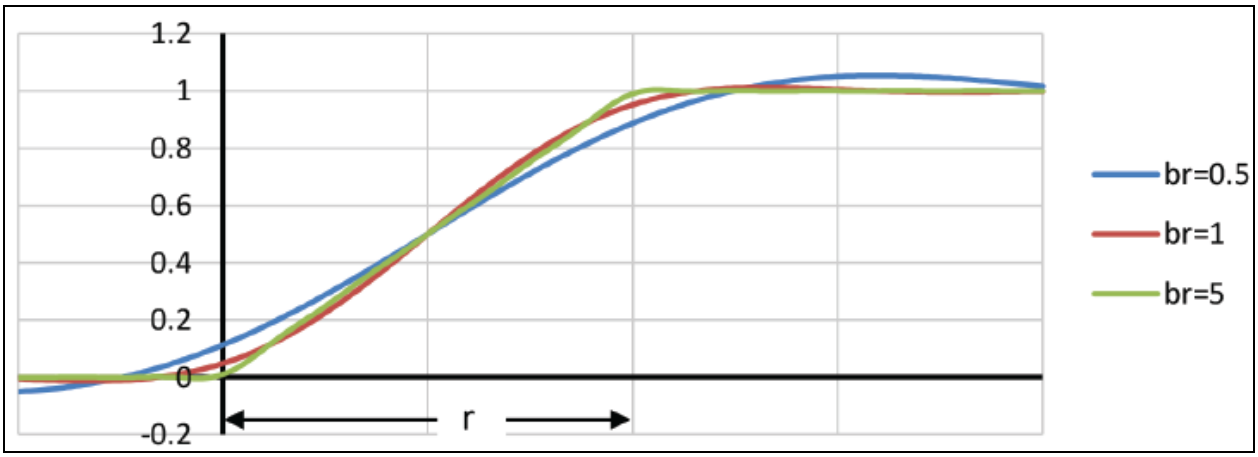


Figure 9. The effect of rectangular windows on edges with rise time *r*. The *y*-intercept and overshoot decline with increasing bandwidth *b*.

Table 2 below quantifies these relationships.

Table 2. Finite edge response for rectangular windows for continuous time

*Bandwidth*Risetime (br)*	*Edge Response at t = 0*	*Max Overshoot*
0.5	0.113	5.34%
0.75	0.056	2.41%
1	0.049	1.19%
2	0.025	0.72%
3	0.017	0.51%
5	0.010	0.32%
10	0.005	0.17%

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As Fig. 9 and Table 2 show, a fairly good finite edge can be achieved with a *br* of 1, but a *br* of around 5 is needed to give a very close approximation to the ideal finite edge.

Impulse Response

The principles behind the step and edge responses also apply to the calculation of impulse responses. Rectangular windows produce the sharpest impulses, but with the greatest amount of oscillation. Hanning and Blackman windows produce impulses that are more spread out, but with less oscillation (Fig. 10).

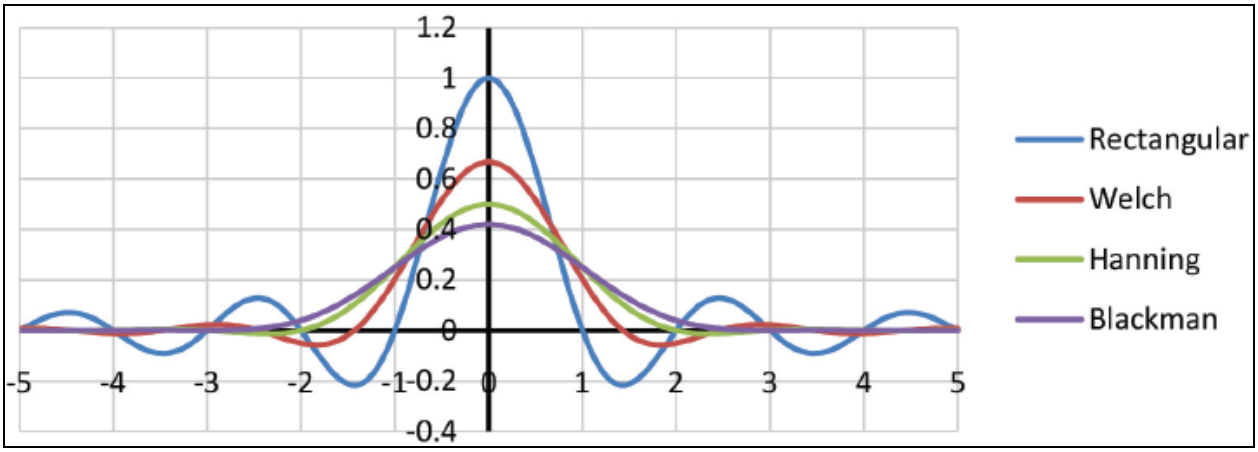


Figure 10. The impulse response for selected windows with a spectral width of 1

Discrete Time Domain Plotting

The preceding discussion treated frequency spectra as continuous functions, but in practice both the frequency and corresponding time data will be discrete. HFSS uses a discrete Fourier transform (DFT) to approximate a continuous time transform, with the frequency step size and upper limit determining the corresponding quantities in the time domain. The default time step and maximum time are given by

$$t_{step} = \frac{1}{2f_{max}} \qquad t_{max} = \frac{1}{2f_{step}}$$

Time resolution is controlled by the upper frequency in the sweep. The maximum time is controlled by the frequency resolution of the sweep. While t_{max} is fixed by the choice of frequency step and cannot be increased after the simulation, t_{step}, or the time delta, can be reduced from the default value within the **TDR Options** dialog box. Decreasing the time delta does not increase the bandwidth of the frequency data, but it does more closely approximate the band-limited continuous time spectra we have so far discussed. Although decreasing the time

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

delta will increase the time required to perform the DFT, the time required is rarely significant. Additionally, a smaller time delta has a significant benefit, as demonstrated in Fig. 11 below.

Fig. 11 shows the step response of a matched lossless transmission line for which the length is controlled by deembedding the driving waveport, using rectangular window functions. The plots on the left are for a short transmission line length and those on the right correspond to a longer length. Fig. 11a shows the time response using the default values for t_{step}. There is some oscillation in the response, which is expected for a rectangular window, but the amplitude of the oscillation is different for the two length cases. This is problematic; since the line is matched and lossless, we expect that a length change will only affect the time delay of the response, not affect the shape or quality of the rising edge. The variation in the response is an undesirable artifact of the coarse time sampling. We can increase resolution by increasing the bandwidth of the sweep, but this requires additional simulation. Fig. 11b shows the same two cases, but with the time delta reduced using the TDR Options dialog box. The results in Fig. 11b agree with our intuition: the edge shape is the same for both line lengths and the only difference is the location of the edge. Setting the time delta to around 1/5 of the default value is generally sufficient, but finer time steps are needed for precise correlation to Tables 1 and 2.

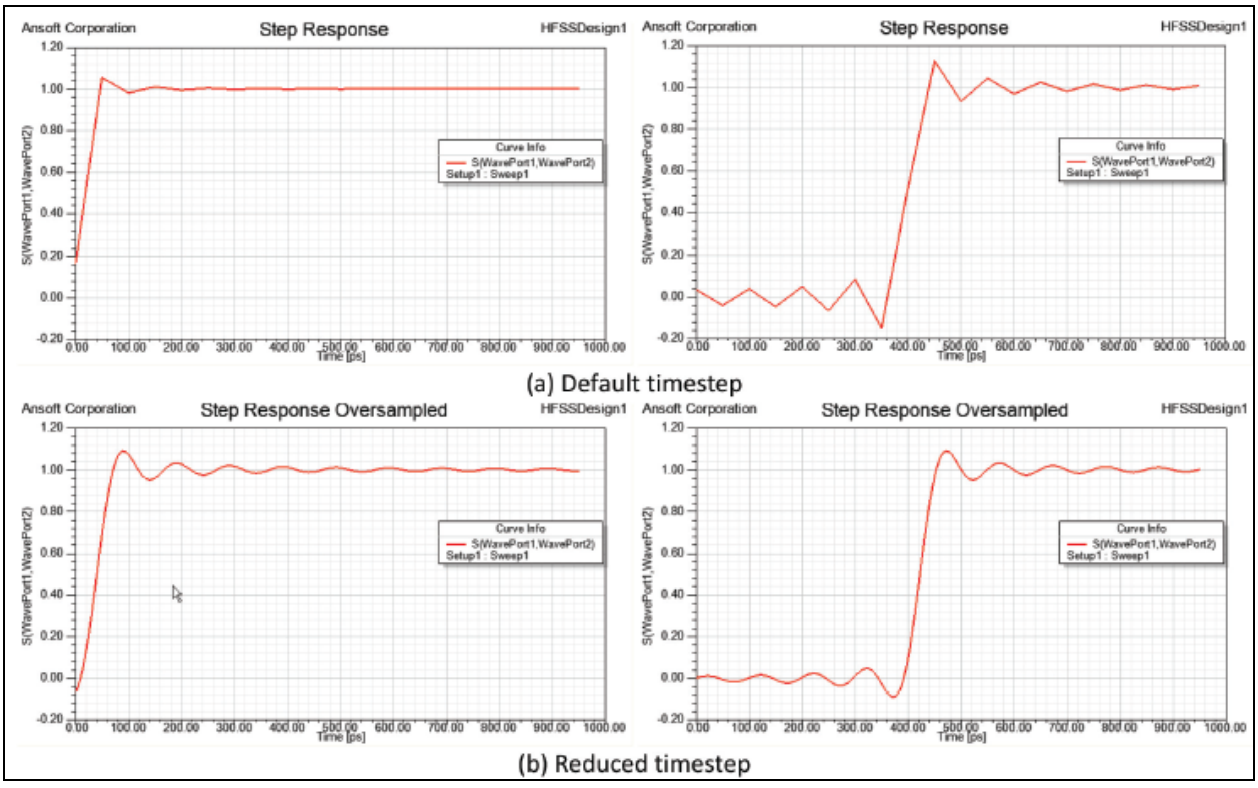


Figure 11. The time domain response of an ideal delay of two different lengths shows that a finer time sampling yields more intuitive results.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The frequency step size governs the length of the time range generated. Although a coarse frequency sampling is often sufficient to generate enough time data for a TDR plot, it is important not to set f_{step} too high in the frequency sweep. Discrete frequency spectra necessarily correspond to periodic time-domain functions, so the calculated step is actually more like a repeating series of long pulses. Fig. 12 shows the how the oscillation decays after the rising edge up to a point, but then begins increasing in anticipation of a falling edge.

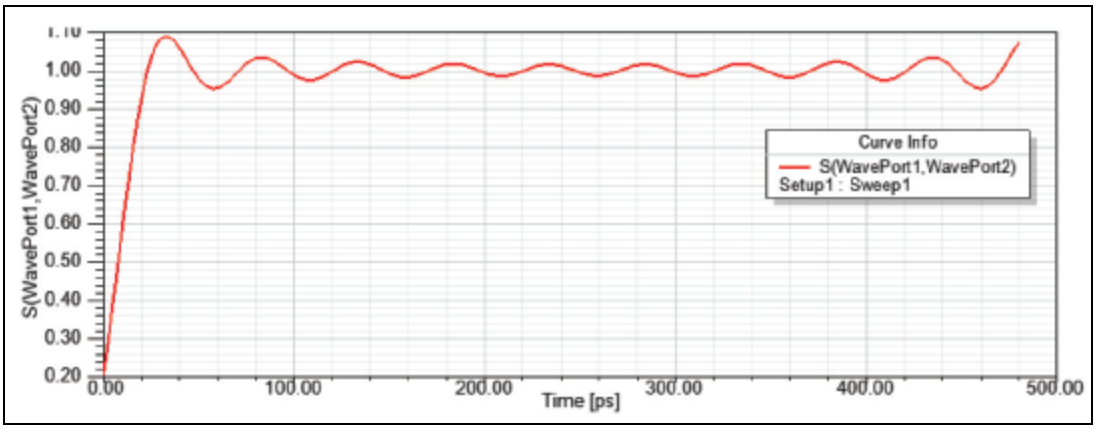


Figure 12. The oscillation caused by a rectangular window eventually starts increasing, due to the periodicity of the waveform

Setting f_{step} to a small value increases the length of the pulse, and minimizes the influence of the future falling edge. Additionally, a smaller f_{step} ensures that resonances and other sharp features in the frequency data are adequately captured. As t_{step} and f_{step} approach zero, the calculated results will converge on the continuous time descriptions given earlier.

Applications

When simulating a TDR plot, we want the fastest edge possible for the bandwidth of our simulation, subject to our preference for oscillation control. Therefore an edge with a rise time of zero is a good choice. Fig. 13 shows TDR plots of a transmission line with several impedance discontinuities. The results for a rectangular and Hanning window with a 20 GHz bandwidth are compared with those for a Hanning window with a 50 GHz bandwidth, which will necessarily be more accurate due to the higher bandwidth, and can be used as a reference. In all cases, the time step was set substantially lower than 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.

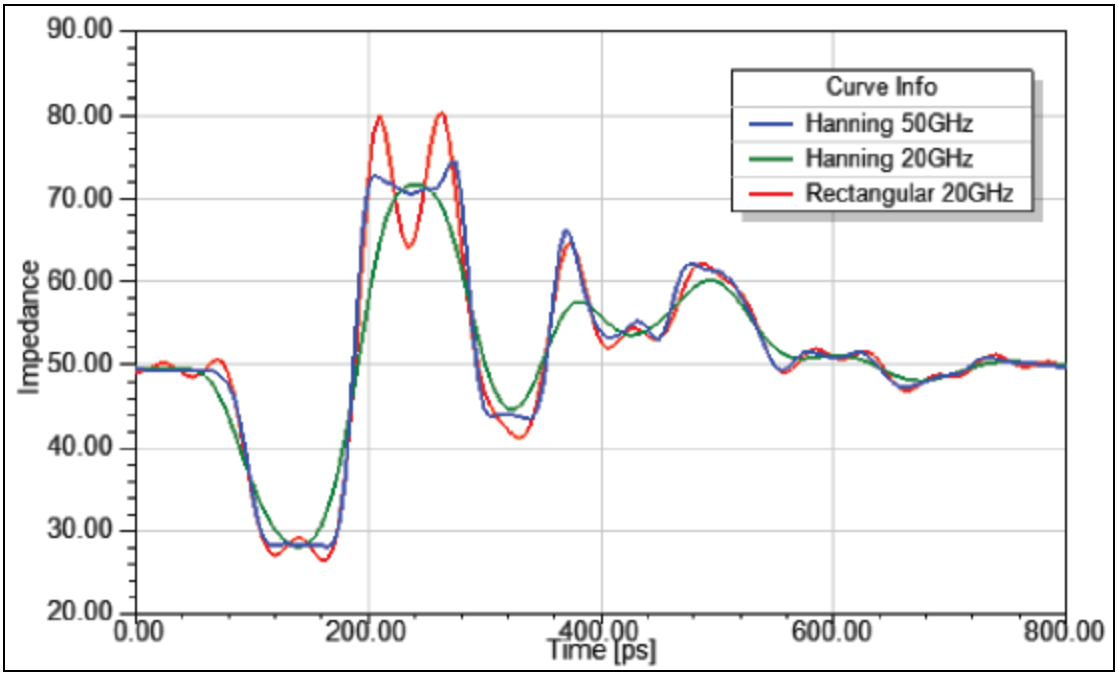
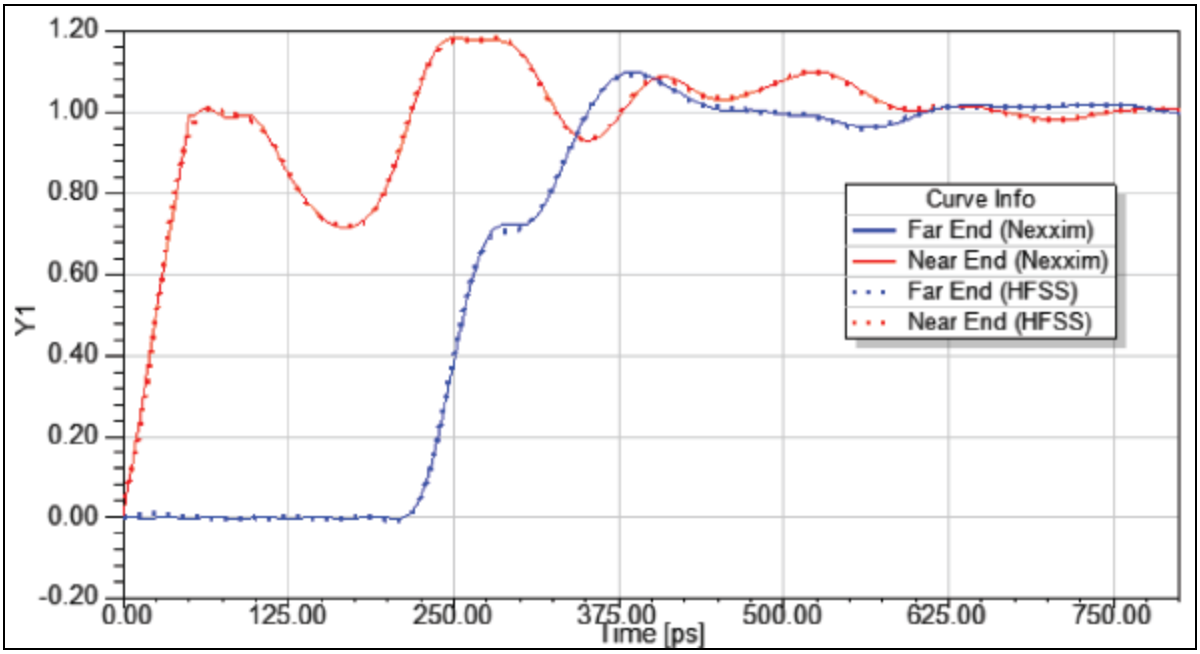


Figure 13. TDR plots for a transmission line with several impedance discontinuities

Fig. 13 shows that the rectangular window effectively captures the sharp impedance transitions, but also displays spurious oscillation. The 20GHz Hanning window does not suffer any oscillation, but gives less resolution on the sharp edges. These results are consistent with the step response characteristics of the different windows we have previously shown.

We can also use time-domain plotting to approximate how a structure would behave in a Nexxim transient simulation. When comparing the results to a transient simulation that uses a pulse or piecewise linear source, it makes sense to use a finite edge with a rectangular window. Fig. 14 compares HFSS and Nexxim results for the transmission line, using a rise time of 50 ps and a rectangular window with a 20 GHz bandwidth *(br* = 1).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



As Fig. 14 shows, very good agreement between Nexxim and HFSS is possible when appropriate settings are used for time domain plotting.

References

Haykin, S., and M. Moher. *Introduction to Analog and Digital Communications*, 2nd ed., Wiley, Hoboken,N.J., 2007.

Kammler, D.W. *A First Course in Fourier Analysis*. Prentice-Hall, Upper Saddle River, N.J., 2000.

Lathi, B.P. *Linear Systems and Signals*, 2nd ed. Oxford University Press, New York, 2005.

Appendix: Window Function Formulas

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rectangular	$\text{Rect}\left(\frac{f}{w}\right)$ where $\text{Rect}(x) = \begin{cases} 1 & -\frac{1}{2} \le x \le \frac{1}{2} \\ 0 & \text{otherwise} \end{cases}$
Bartlett	$1 - \left\lvert\frac{2f}{w}\right\rvert$
Blackman	$0.42 + 0.5\cos\left(\frac{2\pi f}{w}\right) + 0.08\cos\left(\frac{4\pi f}{w}\right)$
Hamming	$0.54 + 0.46\cos\left(\frac{2\pi f}{w}\right)$
Hanning	$0.5\left[1 + \cos\left(\frac{2\pi f}{w}\right)\right]$
Kaiser	$\frac{1}{I_0(\alpha)} I_0\left(\alpha\sqrt{1 - 4\left(\frac{f}{w}\right)^2}\right)$ where $I_0(x)$ is a modified Bessel function of the first kind.
Welch	$1 - \left(\frac{2f}{w}\right)^2$

Animated Reports

The following sections describe how you can postprocess field overlay displays to create and then view various animated reports. Any of the field overlay displays can be animated by cycling the overlays as a series of frames. Refer to the following topics for details on creating the field overlays:

- Overlaying Surface Currents on a 3D View
- Overlaying Far Fields on a 3D View
- Overlaying Near Fields on a 3D View

There are two modes of animation: frequency-based and phase-based. Frequency-based animation creates a series of frames by calculating the field overlay at each of the swept frequencies. Phase-based animation creates a series of frames by calculating the field overlay at a fixed set of phase deviations around a selected frequency. The animation controls can also change the simulation design point.

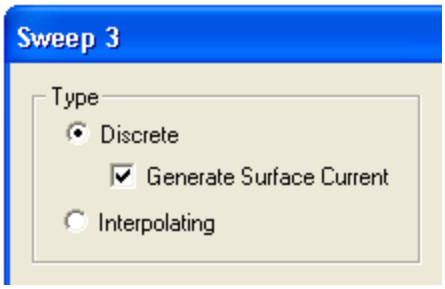
Overlaying Surface Currents on a 3D View

Surface currents calculated as the results of a simulation can be displayed as overlays on the 3D viewer.

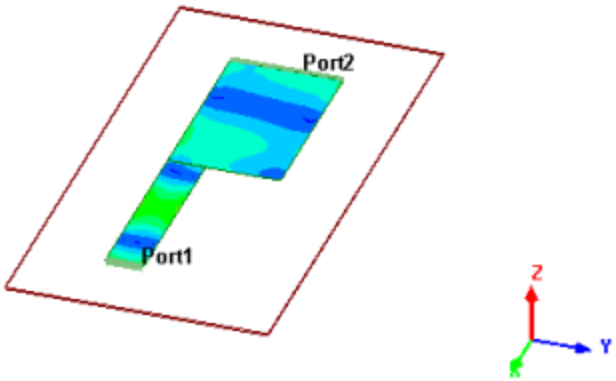
1. To ensure that the surface current information is generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(checked):

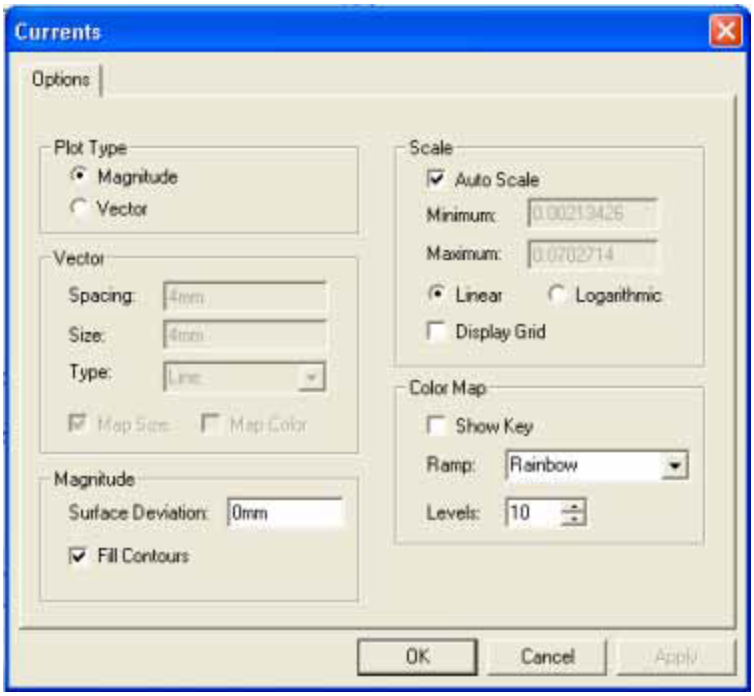


2. Run the analysis with the sweep.
3. To display the surface current as an overlay, expand the **Analysis** icon in the Project window, and select **Setup***m* > **Sweep***n* > **Results** > **Display Currents** (*m* and *n* identify the particular solution setup and sweep setup, respectively). You can also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project** tree. The 3D viewer window appears with the current values overlaid on the geometry:



4. To change the display properties of the surface current overlay, expand the **Results** icon in the Project window, and select **Setup***m*:**Sweep***n*:**Currents***k* > **Properties** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively). The **Currents** dialog opens:

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 **Plot Type** panel, select **Magnitude** to enable the **Magnitude** panel options or select **Vector** to enable the **Vector** panel options.

In the **Scale** panel, select **Auto Scale** (the default), or deselect **Auto Scale** and enter custom **Minimum** and **Maximum** scaling values. Select **Linear** or **Logarithmic** scaling (the default is **Linear**), and toggle **Display Grid** on or off (the default is off).

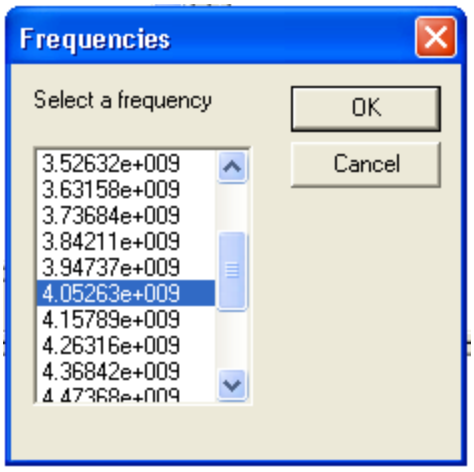
In the **Color Map** panel, select the **Ramp** type (**Rainbow** is the default; other options are **HueScale**, **Magenta**, and **Temperature**), set the number of **Levels** (the default is 10 levels), and toggle the color key (**Show Key**) on and off (the default is off).

Click **Apply** to apply any changes to the display without closing the dialog box. Click **OK** to apply any changes and close the dialog box. Click **Cancel** to close the dialog without changing any options.

5. To select the frequency for the current overlay, expand the **Results** icon in the Project window, and select **Setup*m*:Sweep*n*:Currents*k* > Frequency** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively). The

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequencies dialog opens:



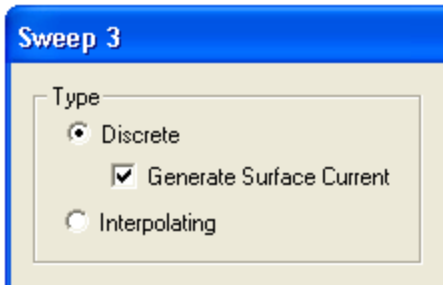
The list displays the frequencies that were swept in the analysis. When you select a frequency from the list, the overlay displays the surface current values calculated at that frequency. Click **OK** to leave the overlay at the selected frequency, or click **Cancel** to close the dialog without applying any frequency changes to the overlay.

6. To dismiss the overlay, expand the **Results** icon in the Project window, right-click **Setup***m*:**Sweep***n*:**Currents***k*, and select **Delete** from the drop-down menu **s** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively).

Overlaying Far Fields on a 3D View

Far fields calculated as the results of a simulation can be displayed as overlays on the 3D viewer.

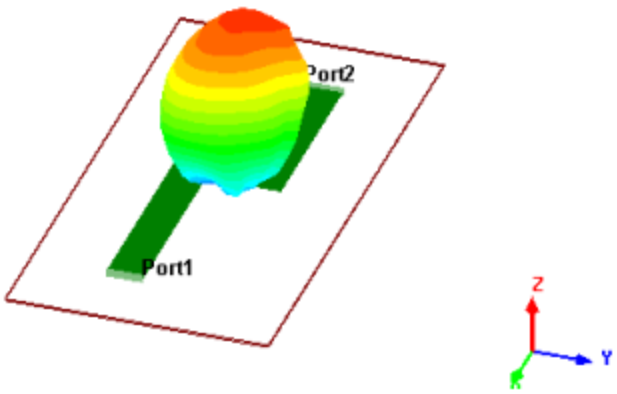
1. To ensure that the far field information can be generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled (checked):



2. Run an analysis with the sweep.

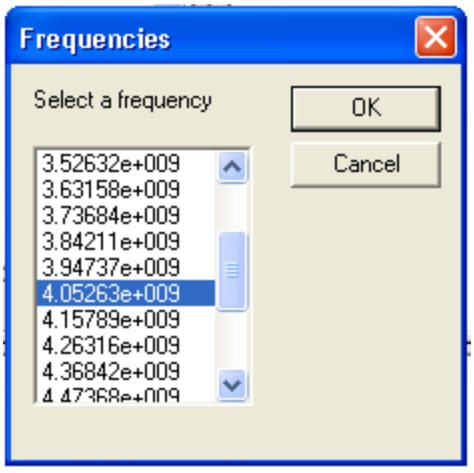
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To display the far field as an overlay, expand the **Analysis** icon in the Project window, and select **Setup***m* > **Sweep***n* > **Results** > **Far Field** (*m* and *n* identify the particular solution setup and sweep setup, respectively). You can also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project** tree. The 3D viewer window appears with the far field values overlaid on the geometry:



The display properties of the Far Field overlay cannot be changed. The Ramp type is Rainbow, and the number of levels is 20.

4. To select the frequency for the far field overlay, expand the **Results** icon in the Project window, and select **Setup***m*:**Sweep***n*:**Far Field***k* > **Frequency**. The **Frequencies** dialog opens:



The list displays the frequencies that were swept in the analysis. When you select a frequency from the list, the overlay displays the far field values calculated at that frequency.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

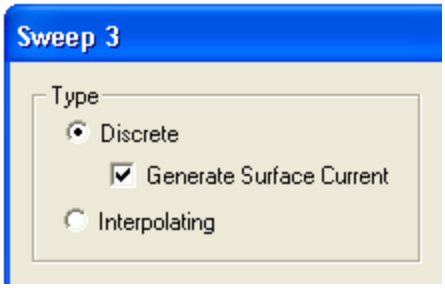
Click **OK** to leave the overlay at the selected frequency, or click **Cancel** to close the dialog without applying any frequency changes to the overlay.

5. To dismiss the overlay, expand the **Results** icon in the Project window, right-click **Setup*m*:Sweep*n*:Far Field*k***, and select **Delete** from the drop-down menu (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and far field setup, respectively).

Overlaying Near Fields on a 3D View

Near fields calculated as the results of a simulation can be displayed as overlays on the 3D viewer.

1. To ensure that the near field information can be generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled (checked):

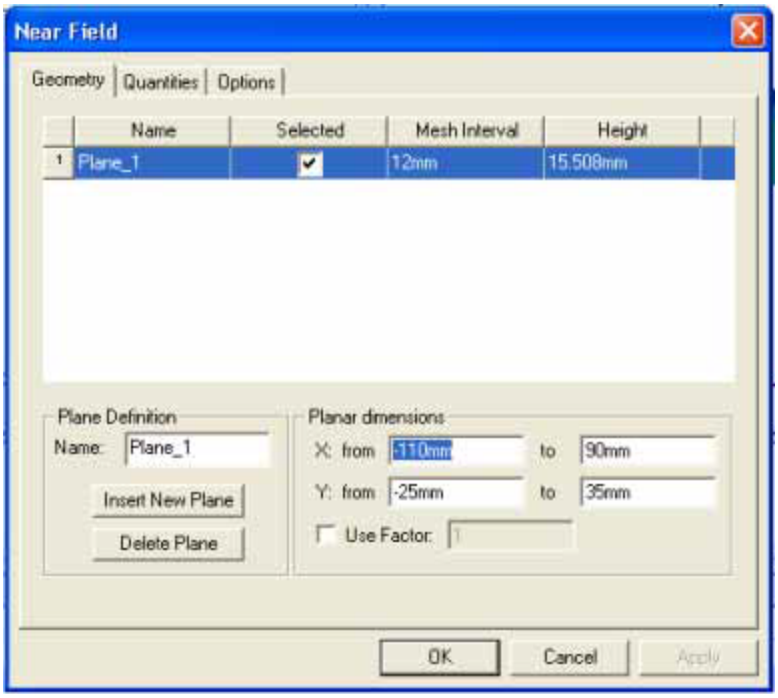


2. Run the analysis with the sweep.
3. To display the near field overlay, expand the **Analysis** icon in the Project window, and select **Setup*m*** > **Sweep*n*** > **Results** > **Near Field** (*m* and *n* identify the particular solution setup and sweep setup, respectively). You can also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project** tree.
4. The **Near Field** dialog opens. The dialog has three tabs, described below. At the bottom of each tab are three buttons:

- **Apply** is activated whenever you change a value. Click **Apply** to start the display, and then to see the effect of each change. The dialog stays open.
- When no values were changed on any tab, the **OK** button starts the display. When one or more values have been changed, **OK** applies the changes. In either case, **OK** closes the dialog and adds an icon for the overlay under the **Results** icon in the Project window.
- **Cancel** is active as long as no changes have been applied. **Cancel** closes the dialog without changing any values. If the overlay is already displayed, it does not change. If **Cancel** is pressed before any overlay is displayed, the overlay is canceled.

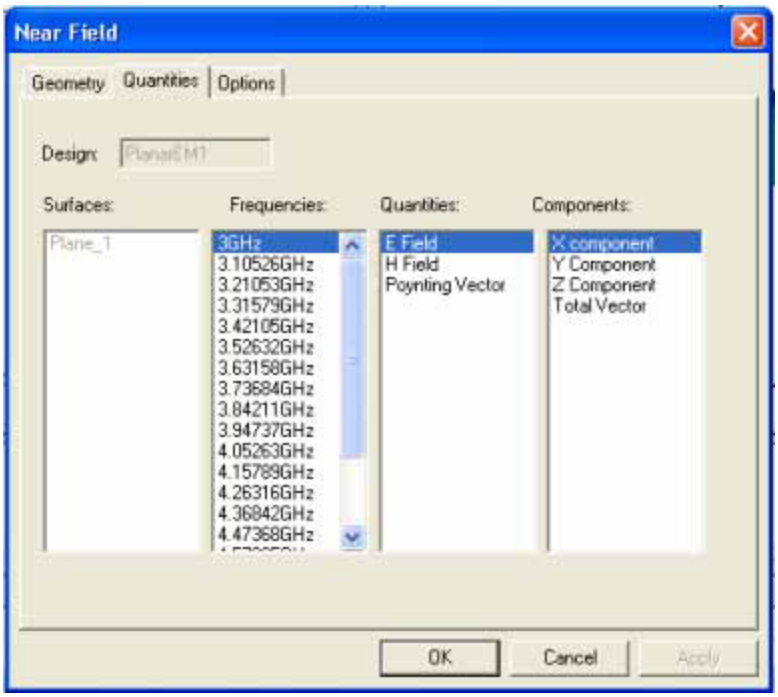
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The **Near Field** dialog opens with the **Geometry** tab displayed:



Use the options in the **Geometry** tab to select (or define) one or more planes for the calculation, including the dimensions to be used, and a scale factor if desired.

Use the **Quantities** tab to select the near field quantity to be calculated:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

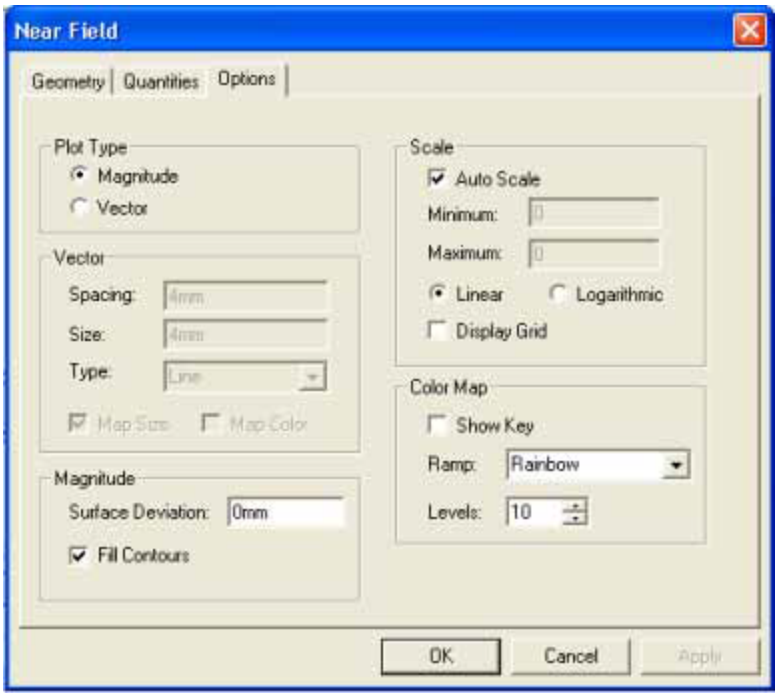
When more than one surface is involved, select a surface from the **Surfaces** list.

Select a frequency from the **Frequencies** list. The frequencies are the ones swept in the analysis.

Select a field type from the **Quantities** list. **E** is the electronic field, **H** is the magnetic field, and the **Poynting Vector** is the (E×H*) field, where H* is the complex conjugate of the H matrix.

Select a vector component from the **Components** field.

Use the **Options** tab to specify the display options for the near field overlay:



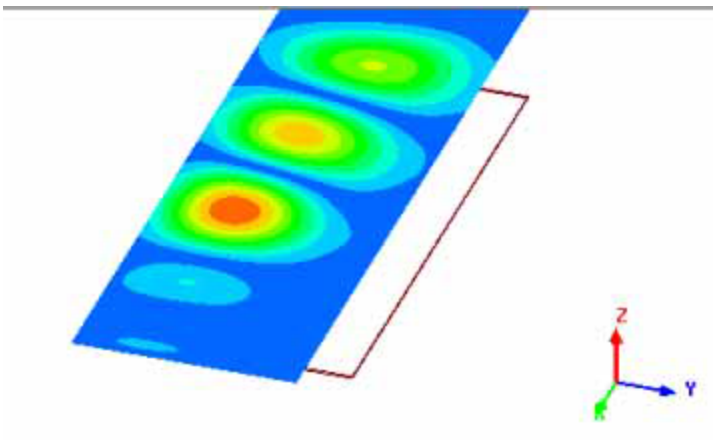
In the **Plot Type** panel, select **Magnitude** to enable the **Magnitude** panel options or select **Vector** to enable the **Vector** panel options.

In the **Scale** panel, select **Auto Scale** (the default), or deselect **Auto Scale** and enter custom **Minimum** and **Maximum** scaling values. Select **Linear** or **Logarithmic** scaling (the default is **Linear**), and toggle **Display Grid** on or off (the default is off).

In the **Color Map** panel, select the **Ramp** type (**Rainbow** is the default; other options are **HueScale**, **Magenta**, and **Temperature**), set the number of **Levels** (the default is 10 levels), and toggle the color key (**Show Key**) on and off (the default is off).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. When you click **Apply** or **OK** in any of the **Near Field** dialog tabs, the 3D viewer window appears with the near field values overlaid on the geometry:



7. To dismiss the overlay, expand the **Results** icon in the Project window, right-click **Setup*m*:Sweep*n*:Near Field*k***, and select **Delete** from the drop-down menu.

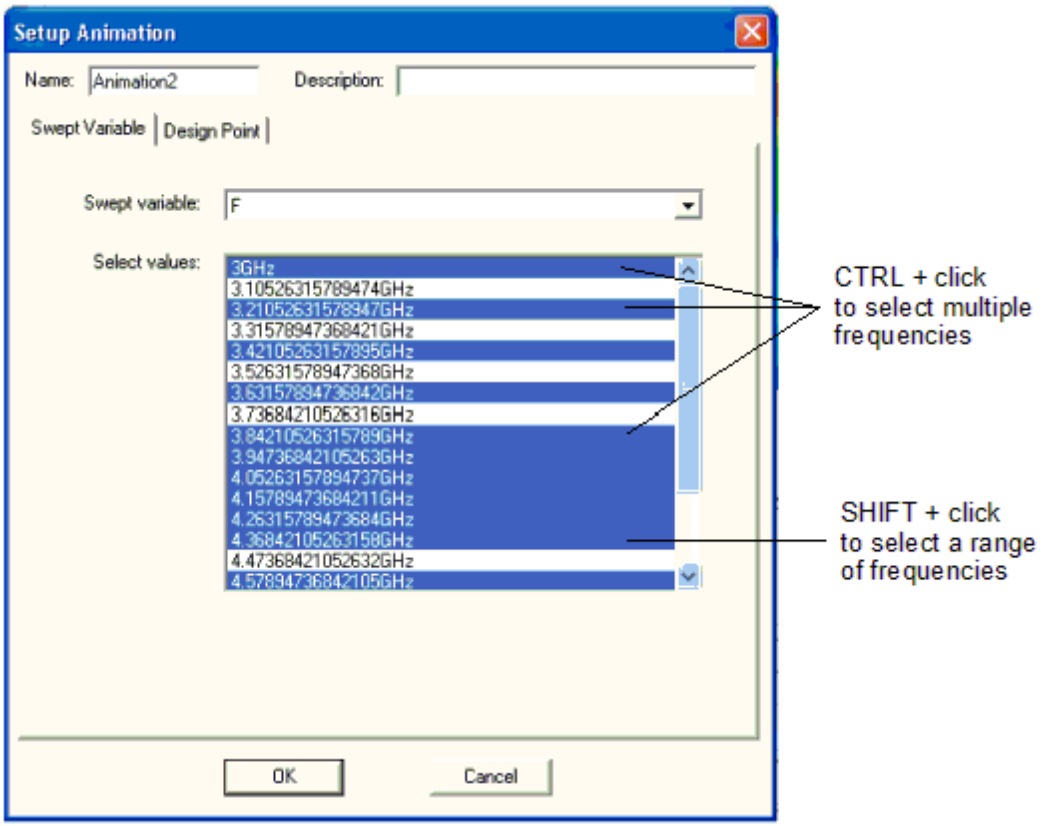
Frequency Animation

1. To initiate the animation of the overlay that is currently displayed, do one of the following:

- Select **Animate** from the **View** menu.
- Expand the **Results** icon in the Project window, right-click the overlay entry, and select **Animate** from 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. If no animations have been defined previously, the **Setup Animation** dialog opens:



- Specify a name in the **Name** field (or accept the default, **Animation***n*, where *n* is a numeral). Optionally, enter a description.
- For a frequency animation, select **F** as the **Swept Variable**.
- By default, all the frequencies are selected (highlighted). Hold down the Ctrl key to select multiple individual frequencies, or hold down the Shift key to select a contiguous range of frequencies. [These selection modes are illustrated in the dialog example above].
- Click **OK**.

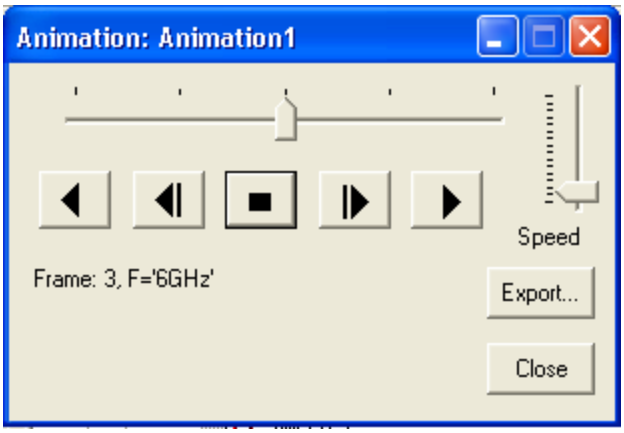
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If one or more animations have been defined, selecting **Animate** from one of the menus opens the **Select Animation** dialog:

The **Select Animation** dialog provides the following operations:

- Click to select one of the animations and click **OK** to start that animation.
- Click **New** to open the **Setup Animation** dialog described above, and close the **Select Animation** dialog.
- Select an animation and click **Delete** to delete the definition.

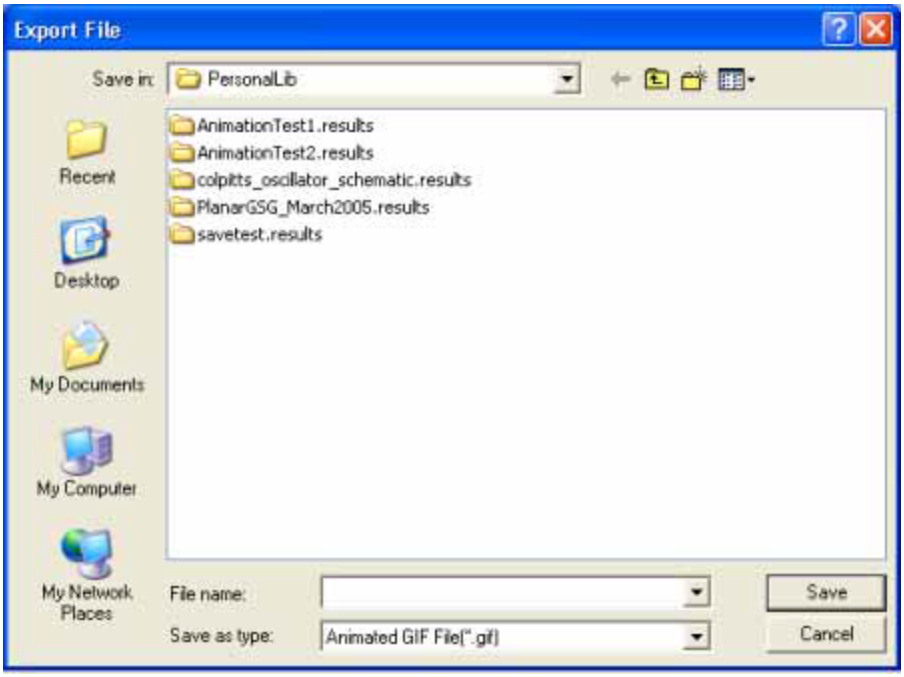
3. The frame data is automatically calculated. If the Progress window is displayed, you can monitor the progress of the calculation. When the frames have been calculated, the animation begins and the **Animation** control panel opens:



- Use the “VCR” buttons to play the animation. From left to right, the buttons are **Reverse**, **FastReverse**, **Stop**, **Fast Forward**, and **Forward**. The indicator at the top of the dialog shows the progress of the animation. Use the **Speed** slider to control the speed of 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.

- To export the frame data to a file, click **Export**. The **Export File** dialog opens:



Specify the directory and file name. Use the **Save as type** menu to select the file format (Animated GIF or AVI). Click **Save** to save the data and close the dialog.

- Click **Close** on the **Animation** control panel to stop the animation and close the panel.

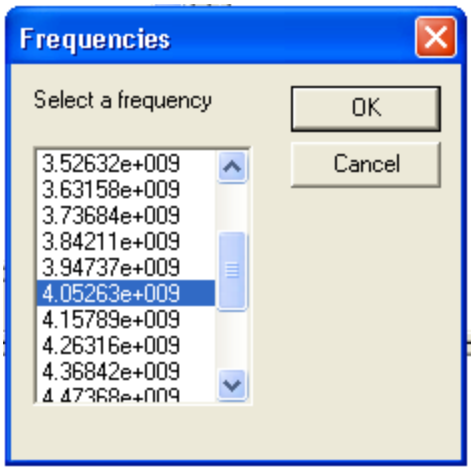
Phase Animation

1. To prepare for phase animation, you must select the base frequency.

- For surface current and far field overlays, expand the **Results** icon in the Project window, right-click the overlay, and select **Frequency** from the menu. The **Frequencies** dialog

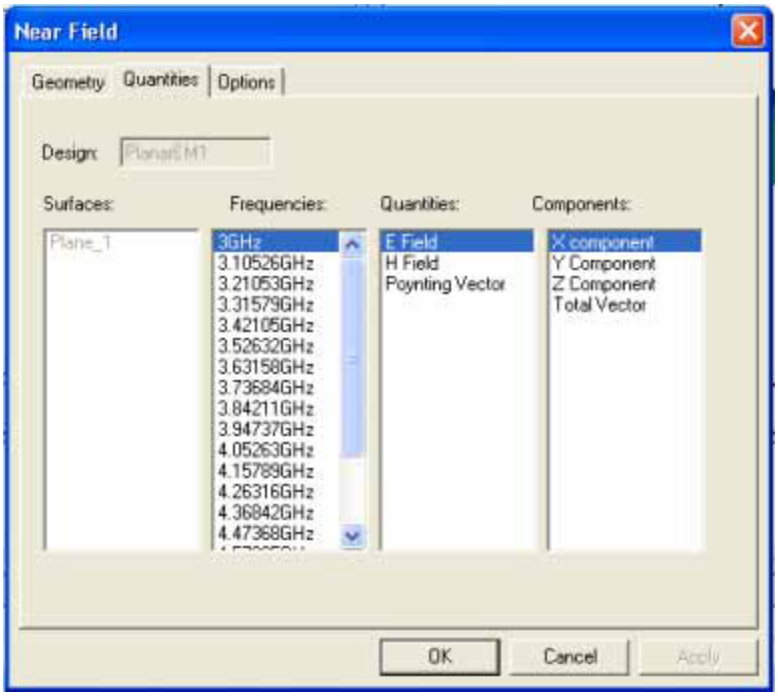
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

opens:



The list displays the frequencies that were swept in the analysis. Select a frequency from the list; the overlay displays the field values calculated at that frequency. Click **OK** to leave the overlay at the selected frequency.

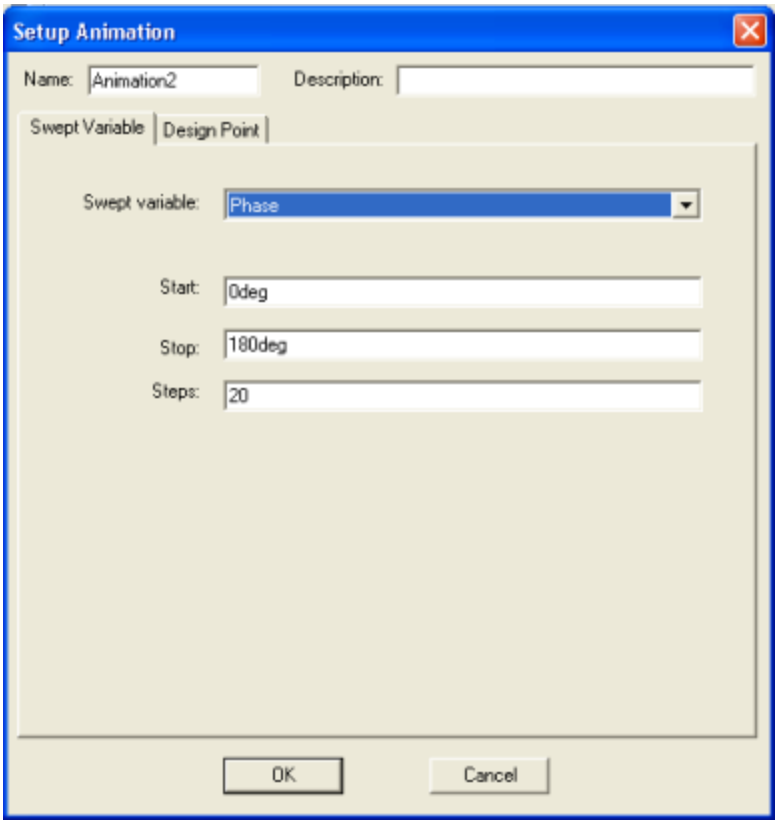
- For near field overlays, expand the **Results** icon in the Project window, right-click the overlay, and select **Properties** from the menu. Use the **Quantities** tab to select a reference frequency for the phase animation:



Click **OK** to apply the frequency and close the dialog.

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 initiate the animation of the overlay that is currently displayed, do one of the following:
   - Select **Animate** from the **View** menu.
   - Expand the **Results** icon in the Project window, right-click the overlay entry, and select **Animate** from the menu.
3. If no animations have been defined previously, the **Setup Animation** dialog opens:



   - Specify a name in the **Name** field (or accept the default, **Animation***n*, where *n* is a numeral). Optionally, enter a description.
   - For a frequency animation, select **Phase** as the **Swept Variable**.
   - Select the Start and Stop phases in degrees.
   - Click **OK**.

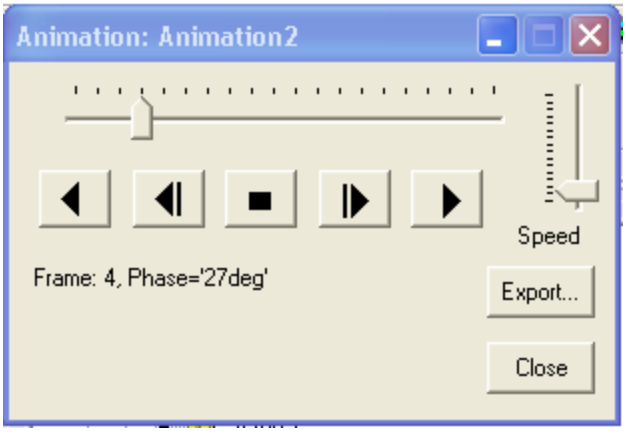
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If one or more animations have been defined, selecting **Animate** from one of the menus opens the **Select Animation** dialog:

The **Select Animation** dialog provides the following operations:

- Click to select one of the animations and click **OK** to start that animation.
- Click **New** to open the **Setup Animation** dialog described above, and close the **Select Animation** dialog.
- Select an animation and click **Delete** to delete the definition.

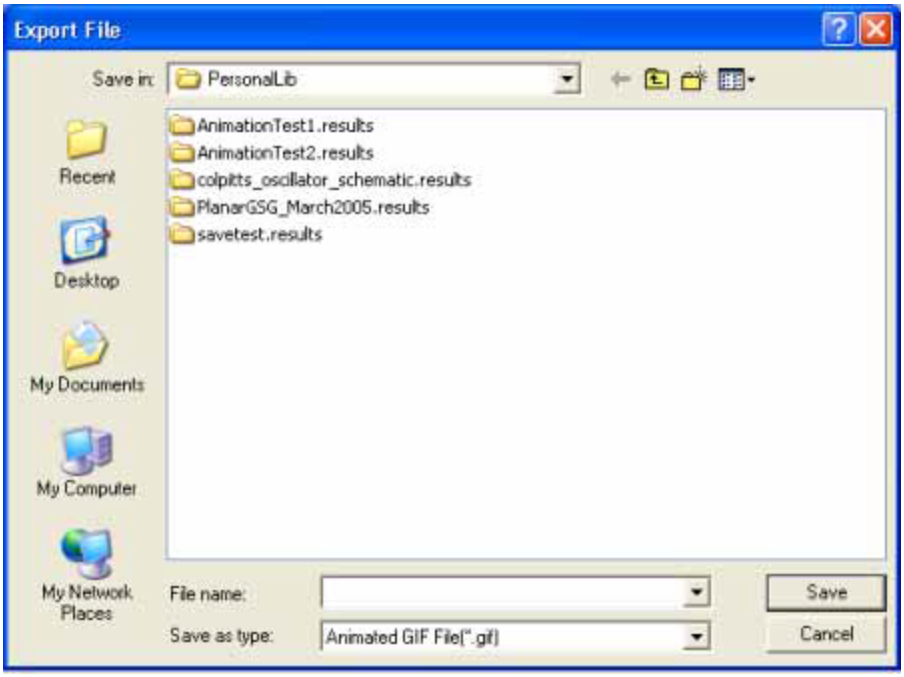
4. The frame data is calculated automatically. If the Progress window is displayed, you can monitor the progress of the calculation. When the frames have been calculated, the animation begins and the **Animation** control panel opens:



- Use the “VCR” buttons to play the animation. From left to right, the buttons are **Reverse**, **FastReverse**, **Stop**, **Fast Forward**, and **Forward**. The indicator at the top of the dialog shows the progress of the animation. Use the **Speed** slider to control the speed of 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.

- To export the frame data to a file, click **Export**. The **Export File** dialog opens:



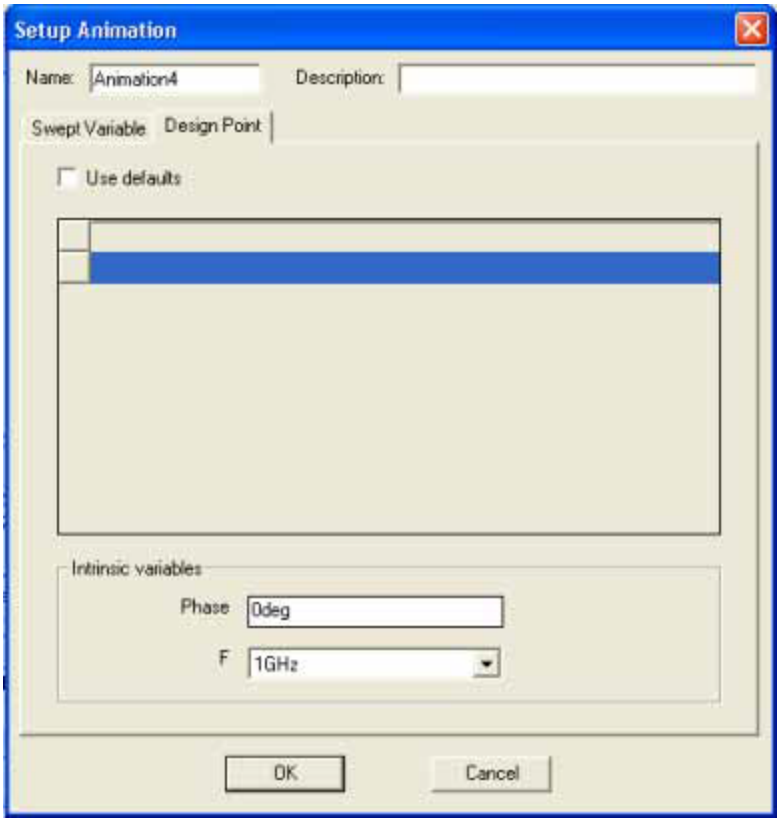
Specify the directory and file name. Use the **Save as type** menu to select the file format (Animated GIF or AVI). Click **Save** to save the data and close the dialog.

- Click **Close** on the **Animation** control panel to stop the animation and close the panel.

Changing the Design Point

Calculating frames for an animation is equivalent to re-simulating the design. You can specify design point parameters for the animation calculations that are different from the ones used in the original simulation. On the **Animation Setup** dialog box, select the **Design Point** tab and deselect the **Use defaults** option. The following fields 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.



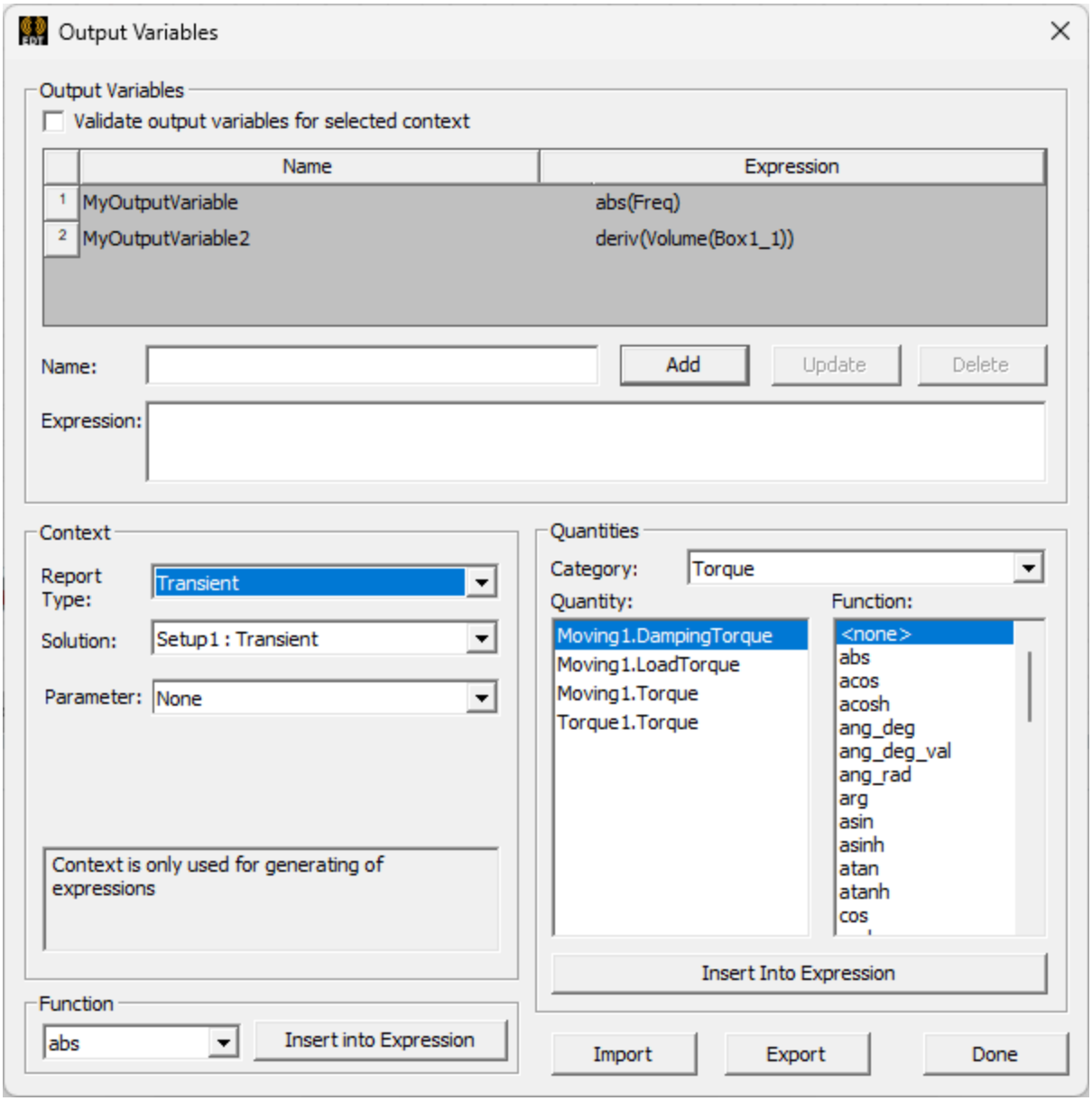
Make any desired changes, and then click **OK** to apply the changes and close the dialog box. Clicking **Cancel** closes the dialog without making any changes.

Specifying Output Variables

Access the **Output Variables** window one of several ways:

- Click **HFSS 3D Layout** > **Results** > **Output Variables**.
- In the Project Manager, right-click **Results** and select **Output Variables** from the shortcut menu.
- Click the **Output Variables** button in the **Report** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Output Variables** window contains four sections:

- **Output Variables** – contains a list of existing output variables, and allows you to specify the name and expression for a new output variable. The **Validate output variable...** check box allows you to validate an expression before adding it. Valid expressions appear in blue and invalid expressions appear in red.
- **Context** – specifies the Report Type and Solution. Depending on the type of report selected, additional fields may appear. All selections affect the functions and quantities 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:**
>
> 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.

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 HFSS 3D Layout, a value from a calculated expression, or an intrinsic variable value (such as frequency or theta).

Important:

Available solution quantity categories depend on the design type, solution type, setup parameters, and plot domain. The selected category provides the default name for the plot, such as "S Parameter Plot *n*". You can edit plot names in the project tree and plot header text in the Properties window.

To select solution quantities to plot:

1. Click **Results** > **Standard Report** and select the type of report you would like to generate.

   The **Report** window appears.

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 **Trace** tab, select a **Category** from the list.

Category	Description
Variables	Intrinsic variables (such as frequency or theta), or user-defined project variables (such as the length of a quarter-wave transformer).
Output Variables	User-defined expressions applied to derive quantities from the original field solution.
S-Parameter	S-parameters from the S-matrix. For designs which include a Frequency Selective Surface (FSS)-referenced radiation boundary, S_{11} and S_{21} represent the extracted reflection and transmission coefficients, respectively.
Y-Parameter	Admittance matrix parameters computed from the S-parameters and port impedances.
Z-Parameter	Impedance matrix parameters computed from the S-parameters and port impedances.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Group Delay	Quantity calculated as rate of change of the total phase shift with respect to angular frequency: $\frac{d(\Phi)}{d(\omega)}$ Group delay is a measure of the time delay of a signal's envelope as it passes through a system. In the context of scattering (S)-parameters, it tells the user how much of a delay a signal experiences at a specific frequency when traveling through a network (i.e., a filter, transmission line, or an amplifier.) **Group Delay Definition (S-Parameters)** Group delay (τ_g) is defined as the negative derivative of the phase of the S-parameter, with respect to angular frequency (ω): $\tau_g = -\frac{d\emptyset}{d\omega}$ Where: 1. ($\emptyset$) is the phase (in terms of radians) of the relevant S-parameter (i.e. S_{21} for transmission) 2. $\omega = 2\pi f$ is the angular frequency. Alternatively in terms of frequency for (f): $\tau_g = -\frac{1}{2\pi} \bullet \frac{d\emptyset}{df}$ **Example:** If you measure S_{21} over frequency, unwrap the phrase response to compute: $\tau_g(f) = -\frac{1}{2\pi} \bullet \frac{darg(S_{21}(f))}{df}$ This provides a function of frequency that indicates how dispersive the network is (i.e., if different frequency components of a pulse experience different delays).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
VSWR	Voltage standing wave ratio, calculated from the equation: $\frac{1+\lvert S_{ij}\rvert}{1-\lvert S_{ij}\rvert}$
Gamma	Propagation constants for the S-parameters.
Port Z_o	Characteristic port impedances.
Matched S Parameter	Matched S is the S-matrix produced by solver. It becomes useful to view when passive ports exist and the matched S-matrix is rectangular instead of square. When passive ports exist, S is the square matrix of the active ports and the passive ports represent a loss mechanism. Matched S-matrix provides coupling information to the passive ports. In Transient, we show Matched S entries while the simulation is in process which is not clean for regular S when port post processing exists.
Short Circuit Z	Short circuit impedance, calculated as: $\mathrm{ShortCircuitZ}(\mathrm{P1}) = \mathrm{mag}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)$
Loop Inductance	LoopL computes with all other ports shorted, even ports on other nets. These short circuits permit induced currents to flow in nearby nets, meaning LoopL will always be lower than a more rigorous loop inductance computation. $\mathrm{LoopL}(\mathrm{P1}) = \frac{\mathrm{im}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)}{(2 \times \pi \times \mathrm{freq})}$
Loop Resistance	Calculated as: $\mathrm{LoopR}(\mathrm{P1}) = \mathrm{re}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)$
Capacitance	This capacitance calculation leaves other nets open-circuited, whereas a rigorous capacitance calculation would short them to ground, meaning "Capacitance" computed here will always be too low. $\mathrm{C}(\mathrm{P1}) = \frac{\mathrm{im}\left(\frac{1}{\mathrm{Z(P1,\ P1)}}\right)}{(2 \times \pi \times \mathrm{freq})}$
Lambda	Guided wavelength.
Epsilon	Effective permittivity.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
TDR Impedance	Time-Domain Reflectometry (TDR) impedance for non-terminal problems. The idea is to excite a structure with a step function, and inspect the reflections as a function of time. This quantity is available only when the **Time** domain is selected. Selecting the TDR Impedance category displays the TDRZ of every terminal or mode in the ports. The list of available Functions includes those that can operate on the TDRZ values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active S-Parameter	Active S parameters represent the reflection coefficient of one port when another port is excited. HFSS offers passive S-parameters as well as active S-Parameters. If you plot Active S-Parameters, HFSS includes the effects of mutual coupling with other ports or elements. One use is in Optimization, to view how S-parameters change as design parameters change. In this case you may plot Active S Parameters. In such cases they allow you to forecast design behavior in the presence of an ideal feed network. A 0w value is a matched load, but 1w values take that into account. Active S-parameters are given as a linear combination of the regular S-parameters, with some weights 'a', based on the matrix equation $\mathbf{b} = \mathbf{S} \times \mathbf{a}$. In the post-processor, the active S-parameter for port*n* is displayed to the user as ActiveS(P*n*:*m*) where "P*n*" indicates the port number and ":*m*" indicates the mode number. Active S, Y, Z, VSWR are supported only for driven modal projects. Power is proportional to the square of the fields. As a consequence, active S-parameters, which are field related, are proportional to the square root of what you set under Edit Sources. You can check this by setting all power to 0 except for one port of interest where you want to see the Active S-parameter. Then for a neighboring port, change the input power in steps, and monitor how the active S-parameter changes. Keep in mind that this involves complex numbers which complicates things. Given a driven model project with a total of *N*port modes, let a_k denote the complex excitation for the *k*-th mode specified in the Edit Sources dialog box. Also let S denote the computed *N* x *N*scattering matrix. If $a_m \neq 0$ define active-S_m, *m* = 1..., *N* by $$active\text{-}S_m \equiv \sum_{n=1}^{N} S_{mn} \frac{a_n}{a_m} \qquad m = 1, \ldots, N$$ If *S* is renormalized or deembedded, the interpretation is that the stimulations are applied to the renormalized external transmission lines at the plane of deembedding. The other relative active quantities are simply transformations on the active S_m.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active Y-Parameter	$Y_0^{(M)}$ is the port admittance of the *m*-th mode. $active\text{-}Y_m \equiv Y_0^{(m)} \frac{1 - active\text{-}S_m}{1 + active\text{-}S_m} \qquad m = 1, ..., N$
Active Z-Parameter	$Z_0^{(m)}$ is the impedance of the *m*-th mode. $active\text{-}Z_m \equiv Z_0^{(m)} \frac{1 + active\text{-}S_m}{1 - active\text{-}S_m} \qquad m = 1, ..., N$
Active VSWR	Active Voltage standing wave ratio is supported only for driven modal projects. $active\text{-}VSWR_m \equiv \frac{1 + \lvert active\text{-}S_m \rvert}{1 - \lvert active\text{-}S_m \rvert} \qquad m = 1, ..., N$
Passivity	A scalar quantity based on the matrix Q = I - conjugate (transpose(S)) * S For every frequency, the value must be no larger than 1. A uniform renormalization of 50 ohms is performed on the solution data for Passivity checking.
Power	For Composite Excitation Solution types, selecting Power as Category lists Quantities for Accepted (Pacc), Reflected (Pref), and Incident (Pinc) powers/port based on your Edit Sources settings.
Design	Enables you to plot or tabulate properties of objects, such as their volumes.
Expression Cache	The values of expressions listed in the Expression cache of the Solution setup can be plotted, for example, as a function of adaptive pass to monitor their convergence.
Expression Convergence	This is intended to plot convergence, as a function of adaptive pass, of expressions in the Expression cache of the Solution setup. In defining the report, for "Context", select Solution: SetupN:Adaptive Pass. ExprDelta will show the change in the value of the expression as a function of adaptive pass, while ExprGoal will show, for comparison, the convergence goal for this expression, as defined in the Analysis Setup under the panel Expression Cache.

3. Select a **Quantity** from the list.
4. If desired, select a **Function**.
5. Click **Close**.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Some of these functions can operate along an entire curve. These are: min, max, integ, avg, rms, pk2pk, cang_deg and cang_rad.

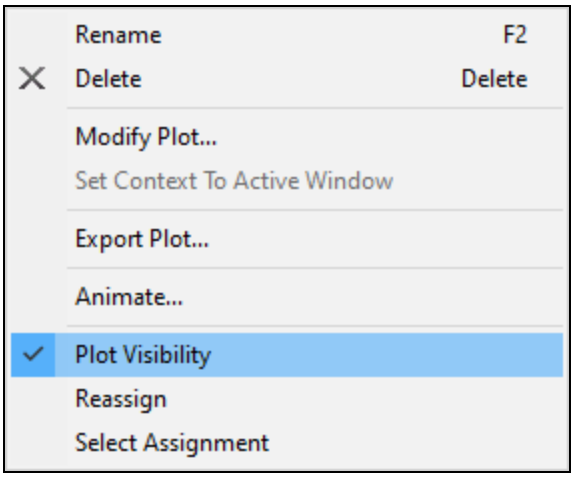
You can select from the functions in the **Output Variables** dialog box's **Function** list or type them directly into the Expression field, if necessary.

Plotting Field Overlays in HFSS 3D Layout

Field overlays are representations of basic or derived field quantities on surfaces or objects for the active design variation. In the Ansys Electronics Desktop (AEDT)™ electronics systems design platform, these overlays display in either the **3D Modeler** window or the **Layout Editor**. Any existing field overlays are listed in the **Project Manager** window, under **Field Overlays**.

To toggle an overlay's visibility:

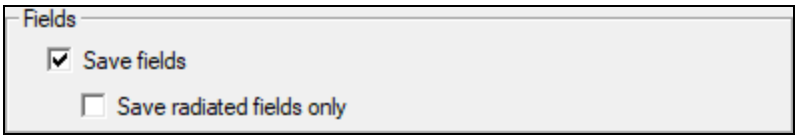
- From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then navigate to the selected field overlay, right-click and select **Plot Visibility**.



Creating Field Overlays

In HFSS 3D Layout, field overlays can be created by selecting combinations of Nets, Layers, and **Clip Planes**. To create a field overlay, follow these steps.

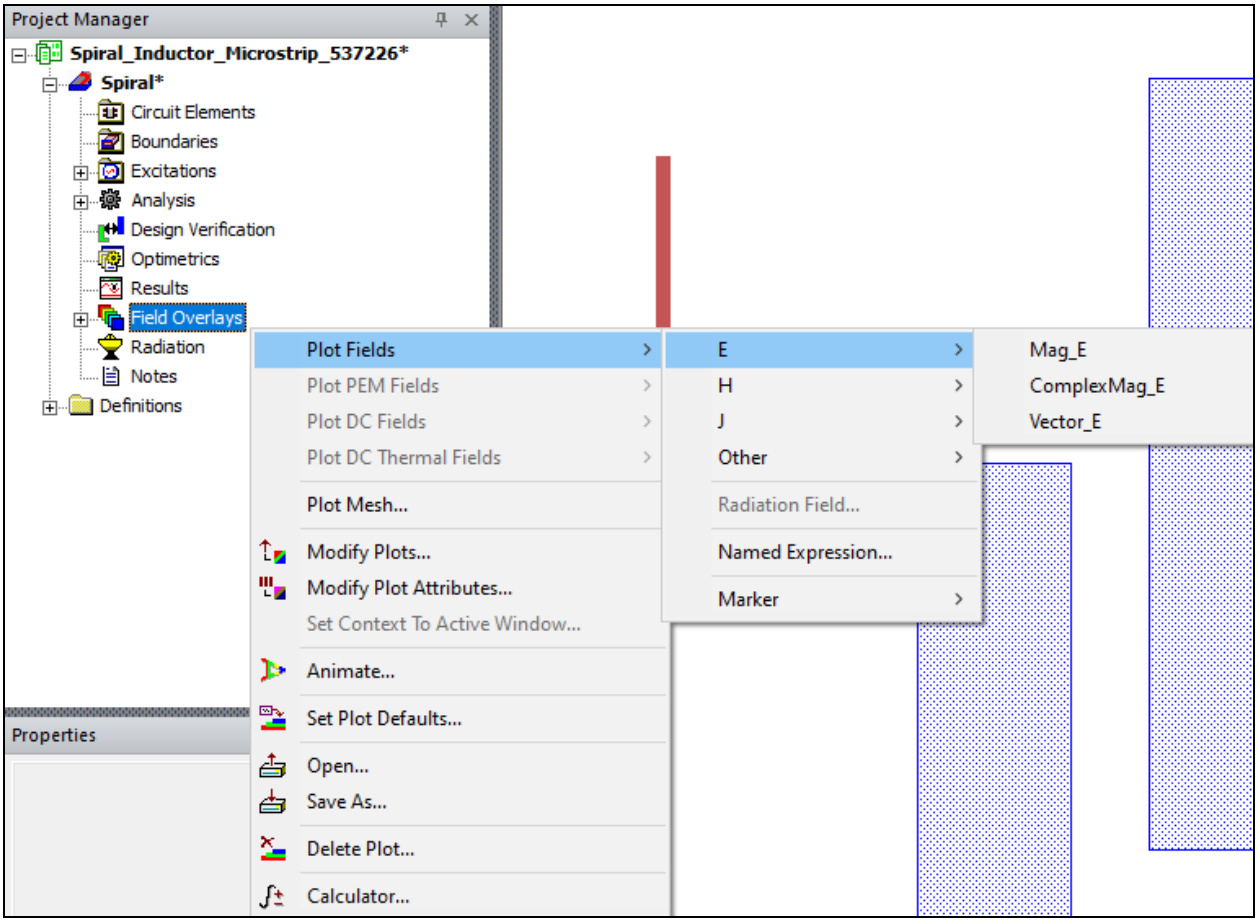
1. **Analyze** an HFSS simulation. During **simulation setup**, ensure the **Save Fields** option is selected.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

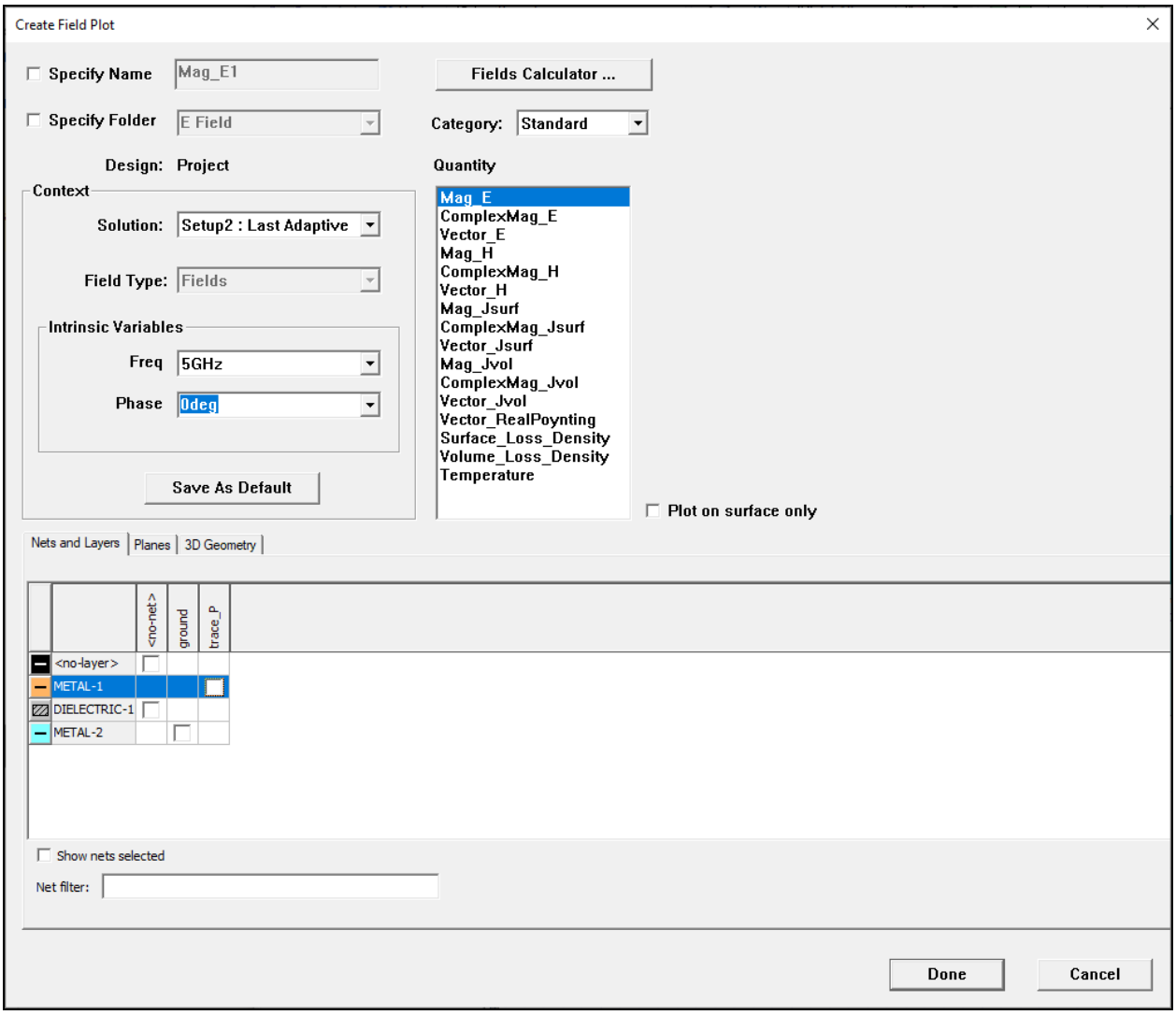
2. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Field Overlays** and select **Plot Fields**. Navigate to the selected field overlay.

> **Note:**
>
> The available overlay options vary, depending on the simulation. For a list of quantities, see: **Field Quantities**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Once a field is selected, the **Create Field Plot** window opens.



In this window:

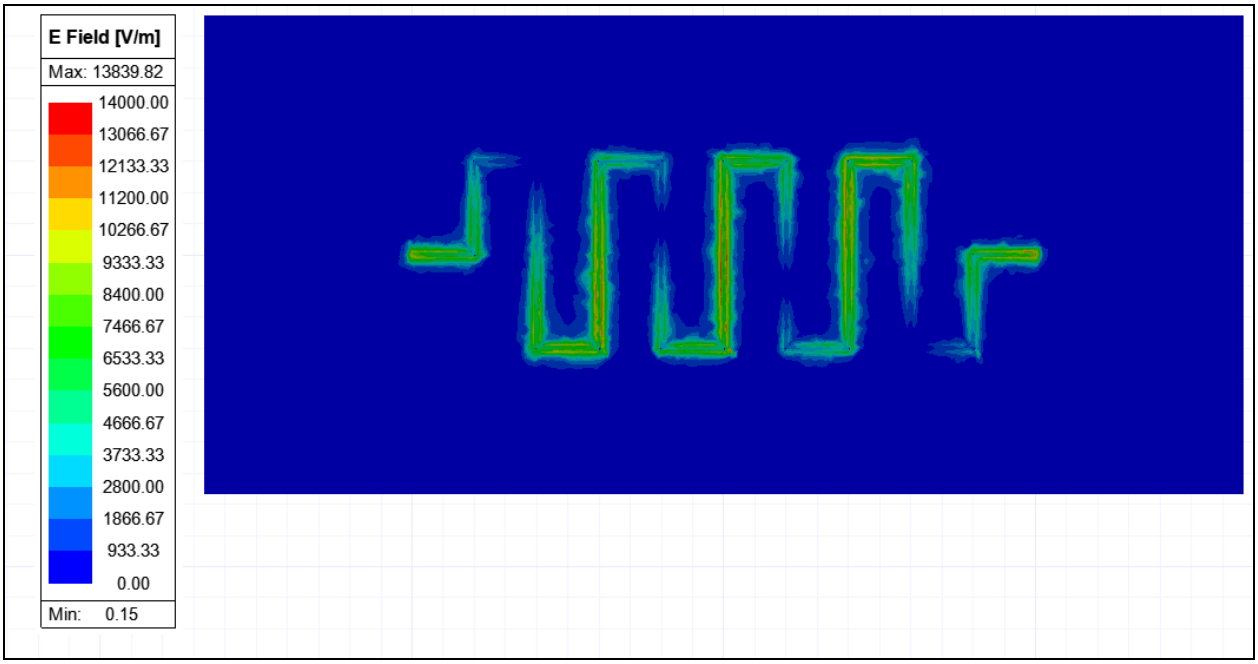
- **Specify Name** – name the field overlay.
- **Specify Folder** – specify the subfolder in which the overlay is placed in the **Project Manager** window.
- **Category** – select **Standard** for default quantities, or **Calculator** to use the **Fields Calculator**.
- ***Context*** Area– choose which **Solution**, **Field Type**, and **Intrinsic Variables** to apply to the field overlay.
- **Quantity** – specifies the quantity being overlaid.
- **Plot on surface only** – select to plot on the surface only.

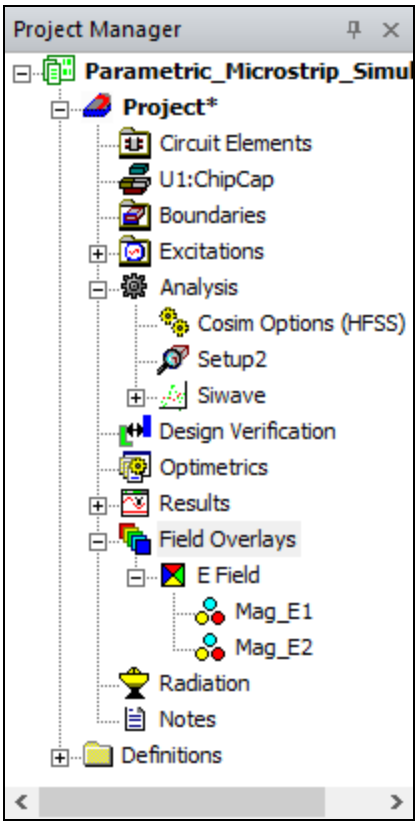
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select Geometry** tabs – on the **Nets and Layers** tab, select nets/layers by checking the appropriate boxes. If the design contains clip planes, an additional **Planes** tab is available, listing the clip planes for the selection. If the design contains 3D geometry, there will also be a **3D Geometry** tab. Use the check boxes to select the nets, layers, clipping planes, and/or 3D geometry on which to apply the overlay.

4. Choose the selected configuration and click **Done**. The field overlay appears in the **Layout Editor**, and is listed in the **Project Manager** window, under **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.





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 field plot is created on a simulation in progress, the field plot is updated after the last adaptive solution. To update the field overlay before the last adaptive solution, select the **Field Overlays** subfolder that contains the field plot of interest, right-click to display the shortcut menu, and select **Update Plots** (e.g., right-click **E Field** to update any **E Field** overlays (**ComplexMag_E**, **Mag_E**, and **Vector_E**)).

Field Quantities

The default field quantities that can be plotted, their definitions, and associated units are as follows:

Field Quantity	Definition	Units
CG Fields		
Q		
SmoothQ	Smoothed surface charge density for CG problems in Q3D.	C/m^2
ABS_Q	Absolute value of SmoothQ.	C/m
E		
Vector_E	The electric field, E(x,y,z,t).	V/m
Mag_E	The magnitude of the electric field, |E|(x,y,z,t).	V/m
DC R/L Fields		
Phidc	Voltage (electrostatic potential) from the DC solution, referenced to the sink terminal of the net.	V/m
VolumeJdc	DC current density vector in the conductor.	$Amps/m^2$
Mag_VolumeJdc	Magnitude of VolumeJdc.	$Amps/m^2$
Volume_Loss_DensityDc	The resistive (ohmic) power loss density from the DC solution, and is defined as: $\frac{1}{\sigma}\left\|f_{\mathrm{dc}}\right\|^2$ Where conductivity of the metal.	W/m^3
DC R/L Thin Conductor Fields		

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

JdcThinCond	DC surface current density vector in a thin conductor.	Amps/m
Mag_JdcThinCond	Magnitude of JdcThinCond.	V/m
PhidcThinCond	Electrostatic potential in a thin conductor.	V/m
AC R/L Fields		
J		
SurfaceJac	Surface current density vector from AC RL solution.	Amps/m
Mag_SurfaceJac	Magnitude of SurfaceJac vector.	V/m
H		
Mag_H	The magnitude of the magnetic field, |H|(x,y,z,t).	Amps/m
Vector_H	The magnetic field, H(x,y,z,t).	Amps/m

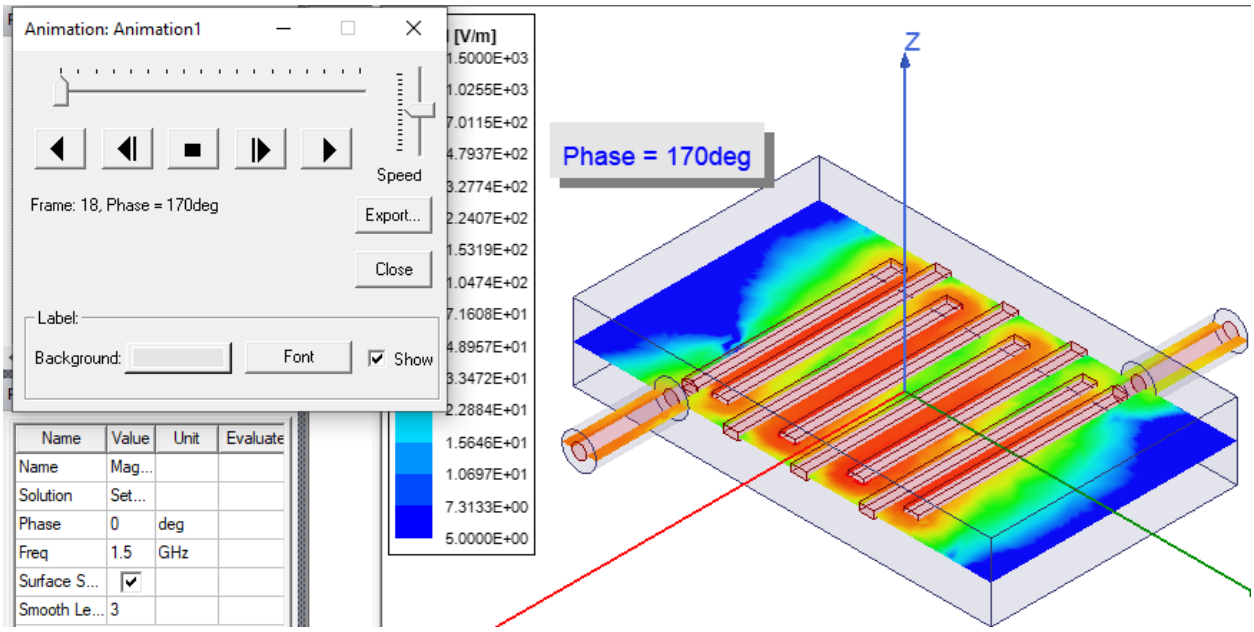
Field Quantity	Definition	Units
HFSS Fields		
E		
Mag_E	The magnitude of the electric field, |E|(x,y,z,t).	V/m
Complex Mag E	The complex magnitude of the electric field, |E|(x,y,z).	V/m
Vector_E	The electric field, E(x,y,z,t).	V/m
H		
Mag_H	The magnitude of the magnetic field, |H|(x,y,z,t).	Amps/m
Complex Mag H	The complex magnitude of the magnetic field, |H|(x,y,z).	Amps/m
Vector_H	The magnetic field, H(x,y,z,t).	Amps/m
J		
Mag Jsurf	The magnitude of the current density, |J|(x,y,z,t), on the surface.	Amps/m
Complex Mag Jsurf	The complex magnitude of the current density, |J|(x,y,z), on the surface.	Amps/m
Vector Jsurf	The current density, J(x,y,z), on the surface.	Amps/m
Mag Jvol	The magnitude of the current density, |J|(x,y,z,t), over the volume.	Amps/m^2
Complex Mag Jvol	The complex magnitude of the current density, |J|(x,y,z), over the volume.	Amps/m^2
Vector Jvol	The current density, J(x,y,z), over the volume.	Amps/m^2

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other		
Vector Real Poynting	The Poynting vector, defined as E x H*.	W/m^2
Surface_Loss_ Density	Resistive power loss density from the AC RL solution.	W/m
Volume_Loss_ Density	Resistive power loss volume from the AC RL solution.	W/m^3
Local SAR	The specific absorption rate.	W/kg
Average SAR	The average specific absorption rate.	W/kg
Certification SAR	The IEEE specific absorption rate certification number.	W/kg

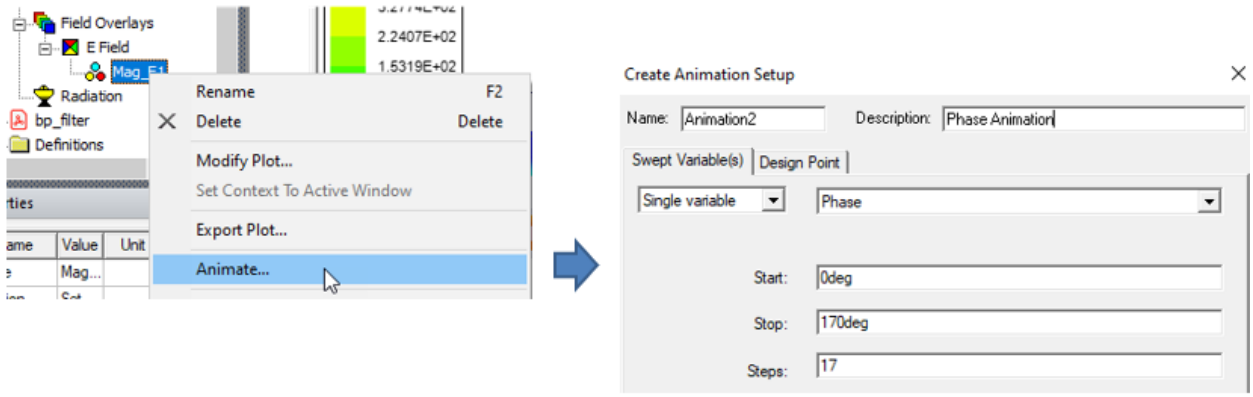
Creating Animations

An animated plot is a series of frames that displays a field, frequency, parametric value, mesh, virtual ray trace or geometry at varying values. Animations can be created in both the General and IC mode environments.



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.

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 describe how to create different kinds of animations.

> **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Phase Animations

You can create phase animations of field plots of frequency sweeps.

Prerequisites

Before creating a phase animation, you must solve a frequency sweep for the project.

Create a field plot.

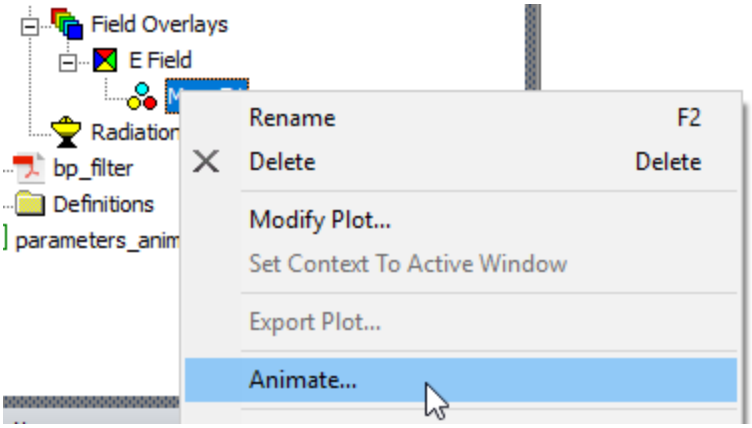
Procedure:

To animate a plot with respect to the phase of the plotted field:

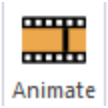
1. Create a field overlay plot to animate.
2. Select the plot in the **Project** tree, and click **HFSS**>**Fields**>**Animate...**, or right-click in the **Modeler** window, and click **View>Animate...**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

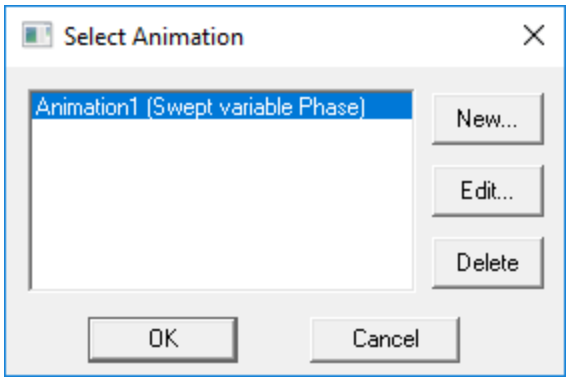
Or right-click the field overlay plot of interest and click **Animate...** from the shortcut menu,



Or select the field overlay plot of interest, select the **View** tab of the ribbon, and click the **Animate** icon.



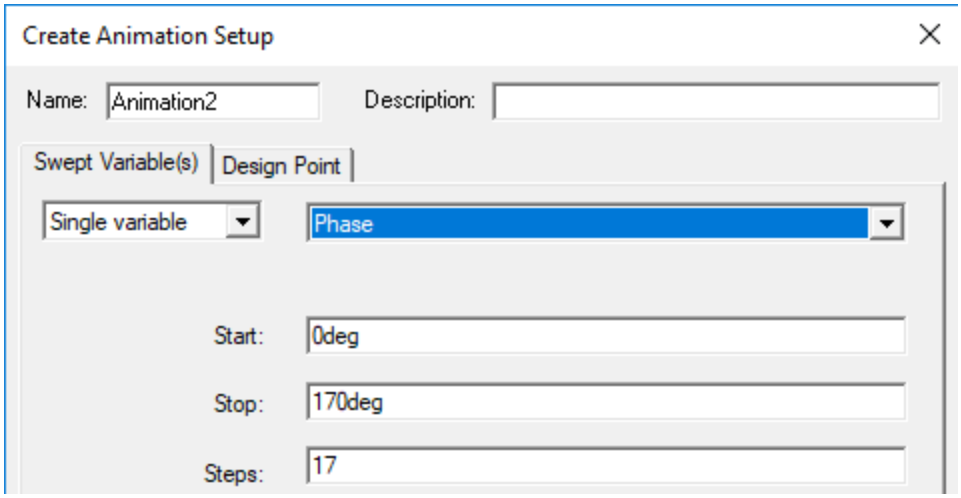
If you already created an animation, the **Select Animation** dialog box appears. Selecting an existing animation from that list starts it.



Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the **Modify Animation Setup** dialog.To create a new animation, click **New**.

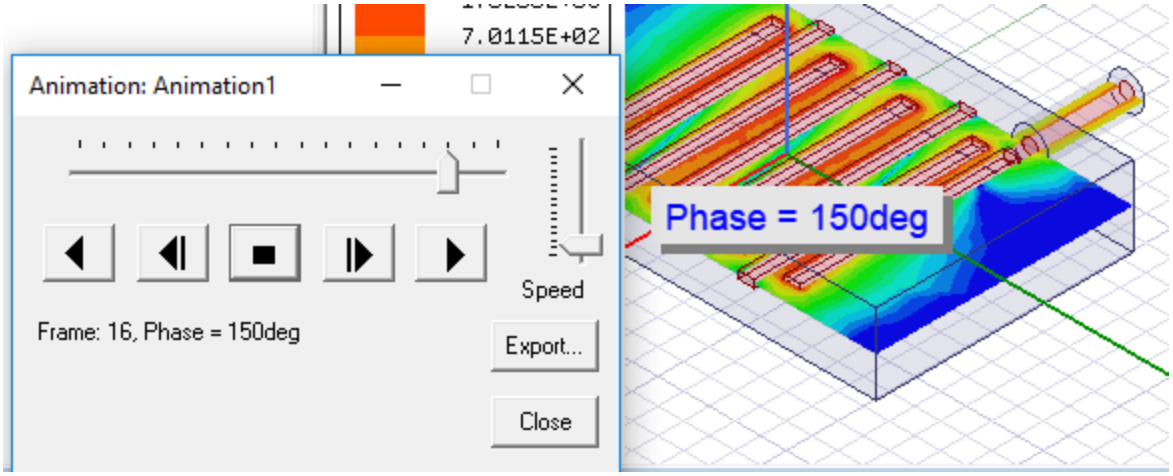
The **Create Animation Setup** dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



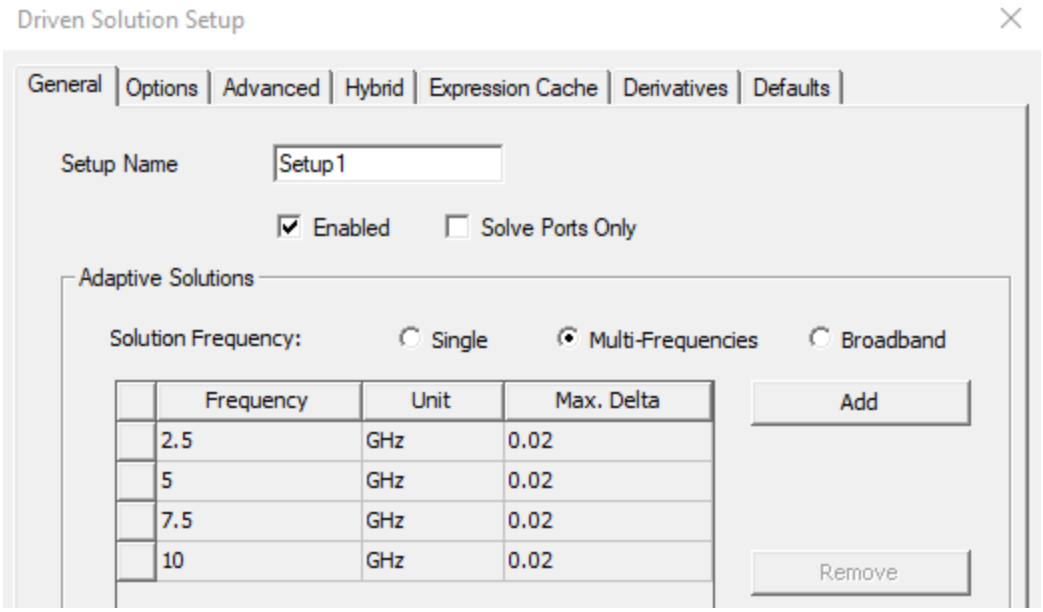
3. Under the **Swept Variable** tab, select **Single Variable** and accept Phase.
4. Specify the phase values you want to include in the animation, as Start, Stop, and Steps to include (e.g., if the **Start** value is **10**, the **Stop** value is **160**, and the number of steps is **10**, the animation will display the plot at 10 phase values between 10 and 160. The start value will be the first frame displayed, resulting in a total of 11 frames in the animation.). If the design has multiple project or intrinsic variables, click the **Design Point** tab to set the values of the non-animated variables.
5. Click the **Design Point** tab.
6. Deselect the **Use defaults** check box.
7. In the table, select the row corresponding to the variable setting of interest.
8. Click **OK** and the animation begins in the view window. The play panel appears in the upper-left corner of the desktop, enabling 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 Frequency Animations

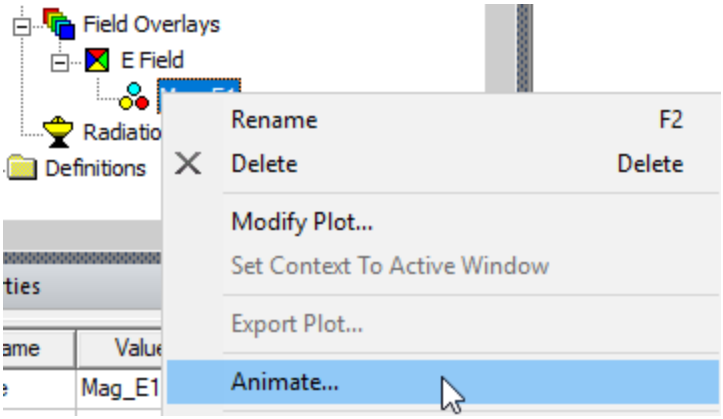
1. In order to create Frequency animations, you must define the frequencies of interest in the Solution setup, using the Multi-Frequencies option.



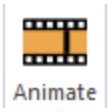
2. After running the analysis, Create a field overlay plot to animate.
3. Do one of the following:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

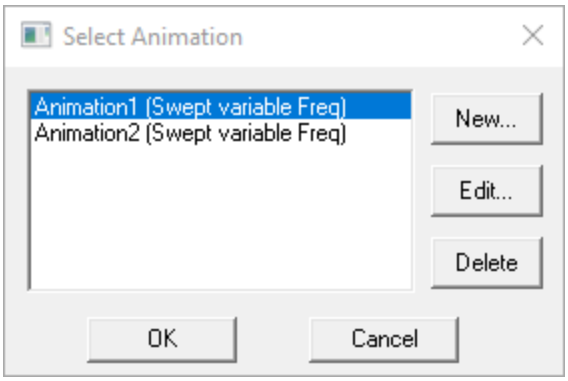
- Click **HFSS>Fields>Animate** .
- Right-click the field overlay plot of interest and click **Animate...** from the shortcut menu.



- Select the field overlay plot of interest, select the **View** tab of the ribbon, and click the Animate icon.

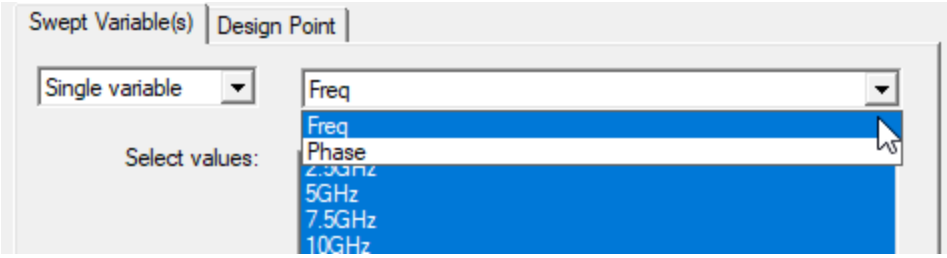


4. If you already created an animation, the **Select Animation** dialog box appears

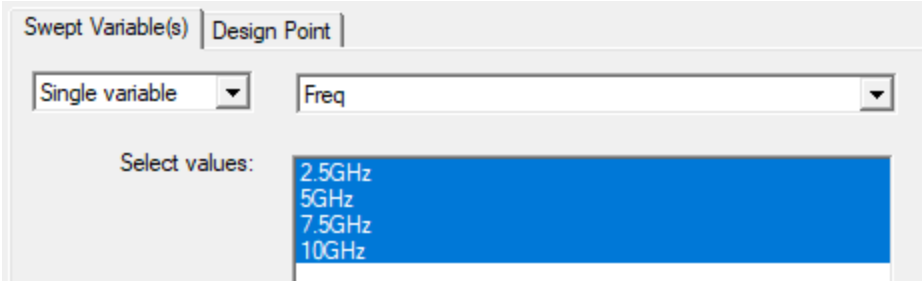


Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the **Modify Animation Setup** dialog. To create a new animation, click **New** and the **Create Animation Setup** dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



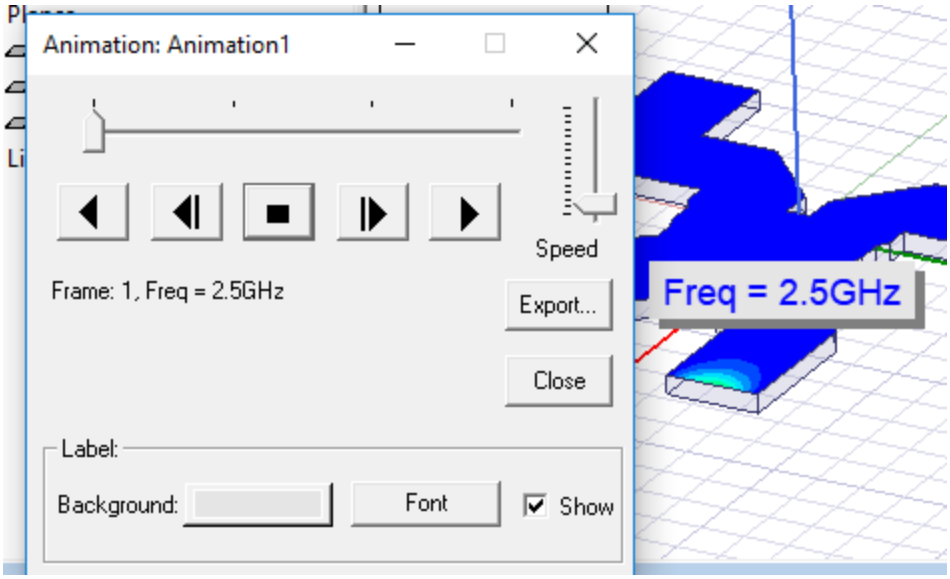
5. Under the **Swept Variable** tab, select **Frequency** from the **Swept Variable** list.
6. Select the frequency values you want to include in the animation from the **Select values** list.



   Use the **Shift** key to select a series of values, and the **Ctrl** key to select values that are not in sequence.
7. If the design has multiple project or intrinsic variables, click the **Design Point** tab to set the values of the non-animated variables.
8. Click the **Design Point** tab.
9. Deselect the **Use defaults** check box. In the table, select the row corresponding to the variable setting of interest.
10. Click **OK** and the animation begins in the view window. It will display one frame for each frequency value you selected. The play panel appears in the upper-left corner 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.

desktop, enabling you to stop, restart, and control the speed and sequence of the frames.



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.

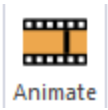
Procedure:

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

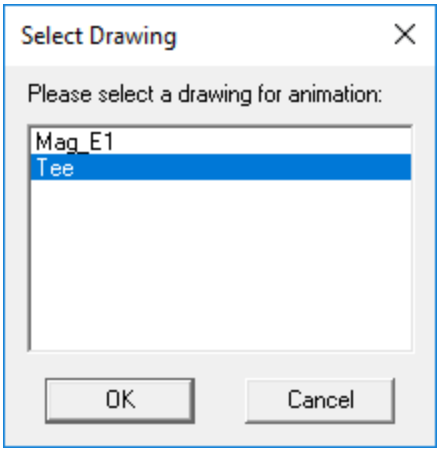
   Select the **View** tab of the ribbon, and click the **Animate** icon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

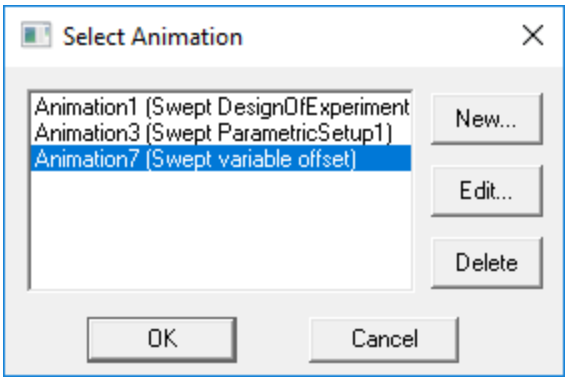


If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

2. In the *Select Drawing* dialog box, select the object that you want to animate.

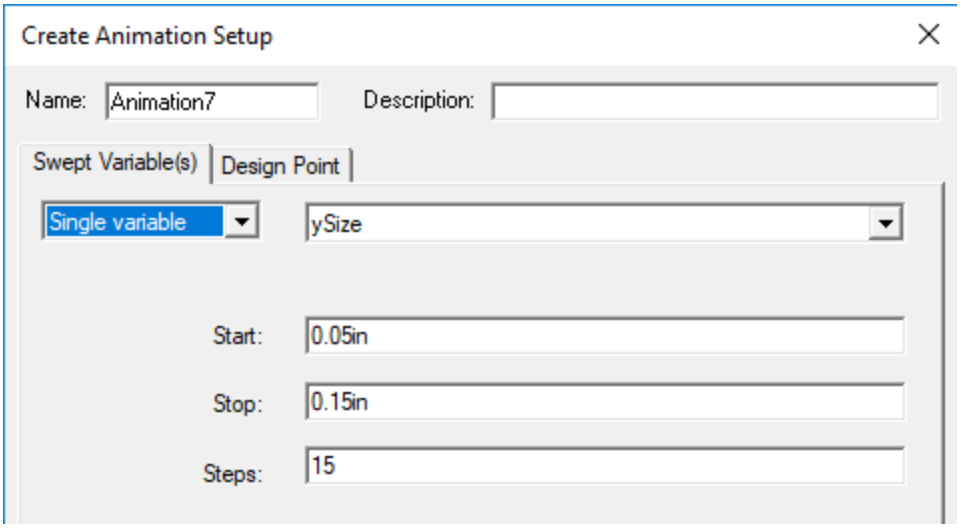


If previous animations have been created for this project, the *Select Animation* dialog box will appear. Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the *Modify Animation Setup* dialog box. You may choose an animation setup from the list 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.

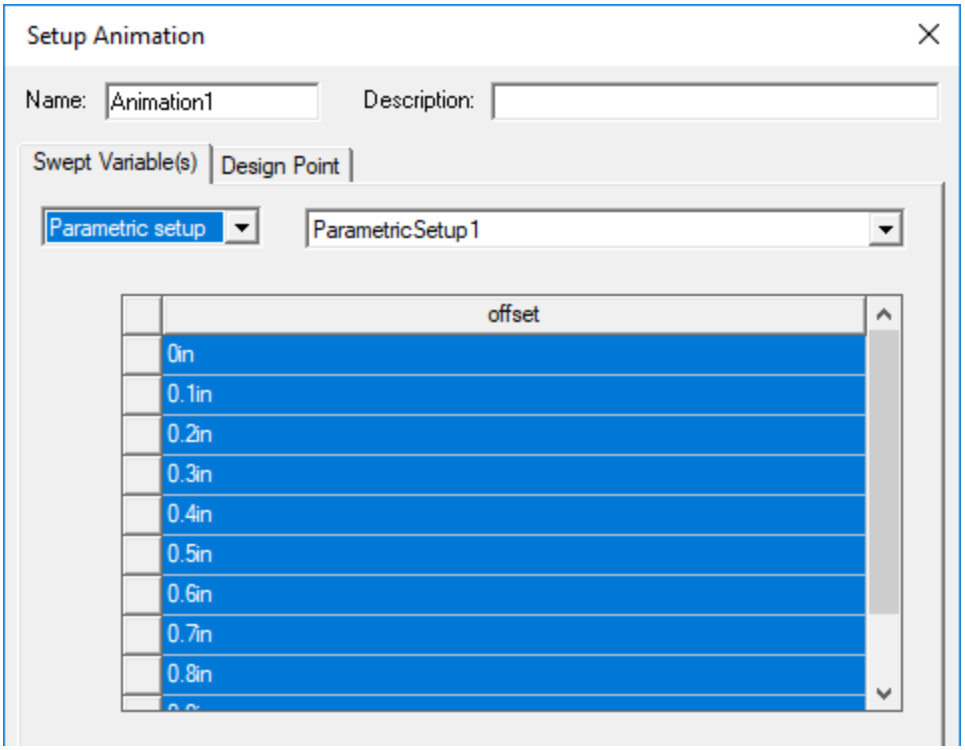


The *Create Animation Setup* dialog box appears:

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.

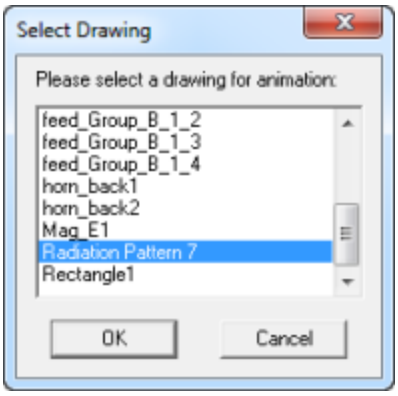
5. If you have created a **Design of Experiments** setup, you can select to define an animation based on that setup and values.
6. If the design has multiple project or intrinsic variables, you use the **Design Point** tab to set the values of the non-animated variables.
7. Click **OK**.

   The animation begins in the view window. It will display one frame for each variable value.

Creating 2D Radiation Field Plot Overlay Animations

In order for a Radiation Pattern plot to be valid for overlay animation, either theta or phi must be set to a single value, so that the pattern lies in a single plane in 3D space. Note that for animation, the plot must contain at least two traces within this plane - for example traces for multiple frequencies of a discrete sweep. You create the 2D Radiation Field plot as before, but must also prepare it to be suitable for animation. See Overlaying 2D Radiation Field Plots on Models.

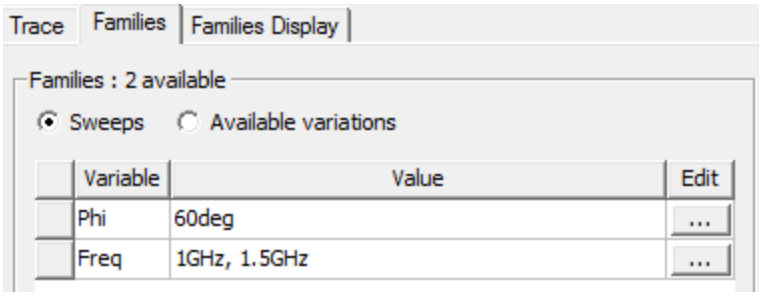
If an existing 2D Plot is a candidate for animation, you will see it listed when you click **View>Animation** to open the **Select Drawing** dialog.



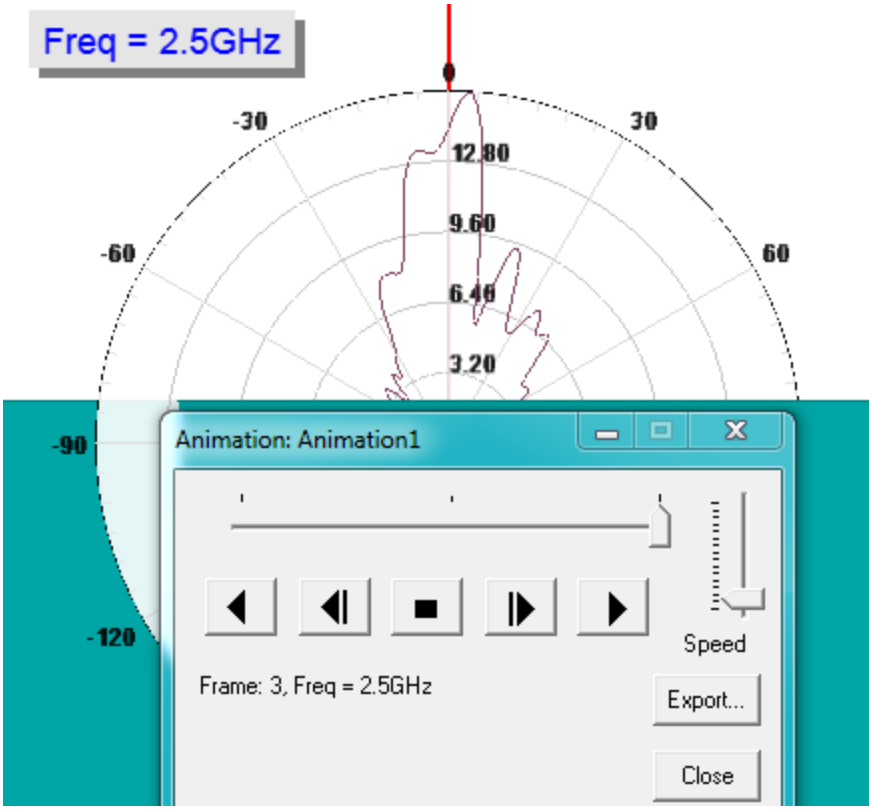
However, if the Radiation Pattern plot does not have at least two traces, when you try to set up the animation, you will receive a notice that you need to Modify the plot or simulation setup appropriate or create a new one.

Recall that the Context section on **Modify Plot** may let you select Setup*n*: Sweep, which will permit you to select additional frequencies. Also the Trace tab and Families tab lets you specify a single Phi and/or Theta and select multiple values for other available Variables 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.



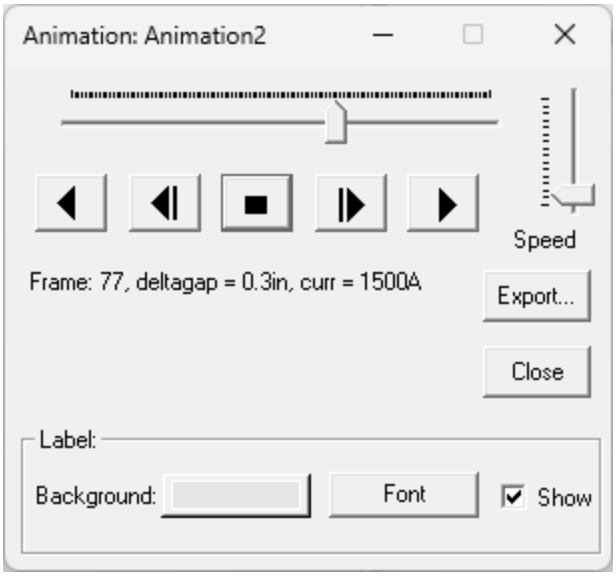
For plot with multiple traces, the animation proceeds through the existing traces.

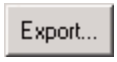


Controlling the Animation's Display

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.

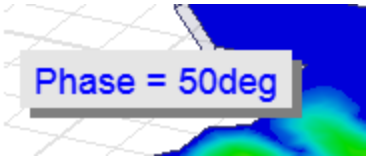
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 information	The current frame and phase at which the plot is being displayed is listed below the control buttons.
	Enables you to export the animation to an animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format.
	Closes the animation window.
Show label	If you select the Show check box, a label showing the swept variable value

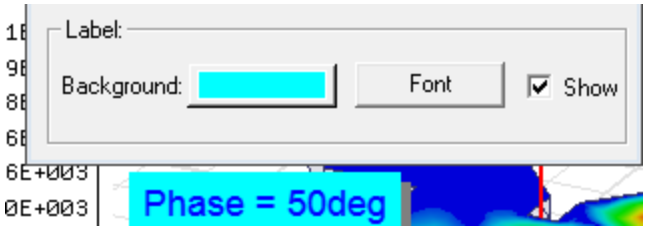
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

appears in the 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

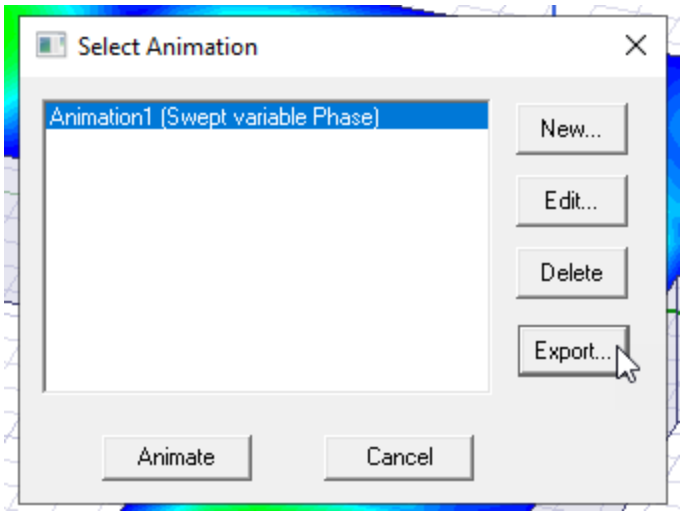
The font button opens a font selection dialog that you can use to set the Font, Font Style, and Size for the label. The Default is Arial Narrow 14pt.

Exporting Animations

You can export animations from the **Animation** control dialog box, or, for Ensight format, from the **Select Animation** dialog box.

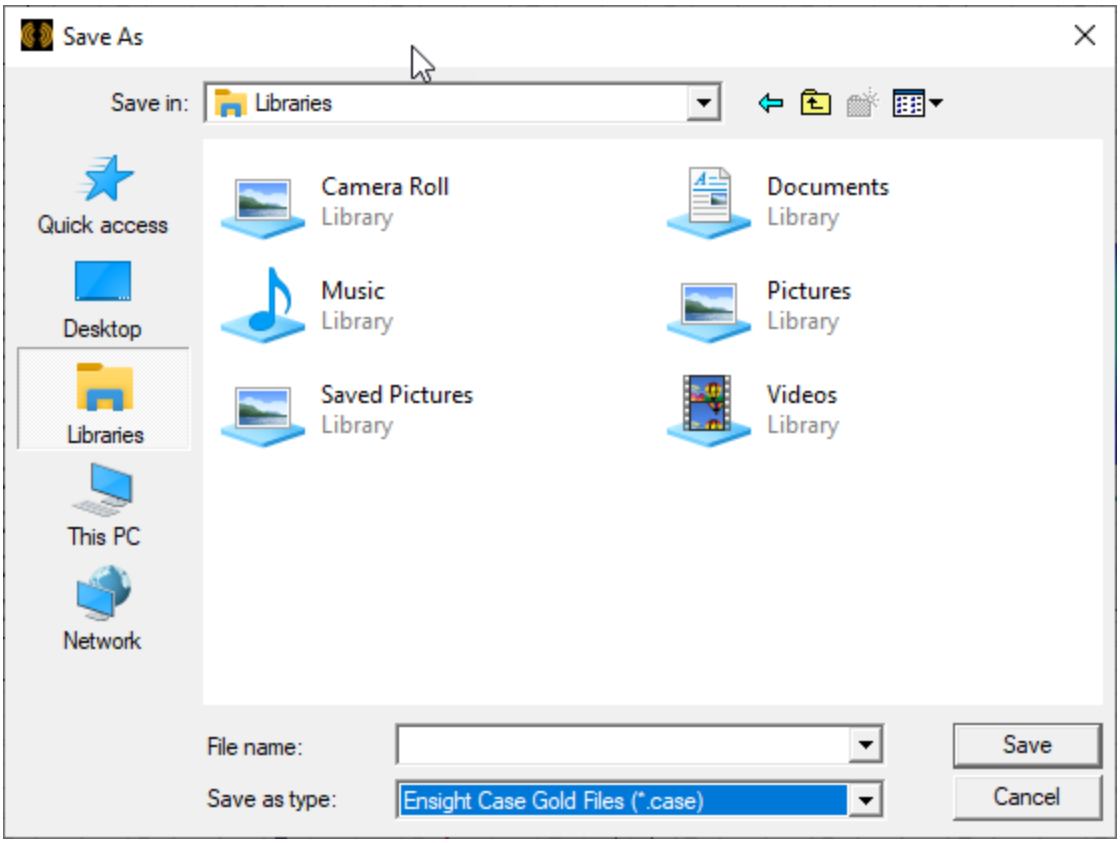
Exporting Ensight from the Select Animation Dialog Box

1. Create the animation you want to export.
2. In the **Select Animation** dialog box, select **Export...**



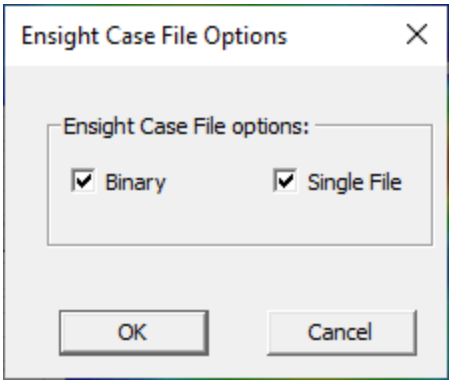
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. This opens the Browser window.



Navigate to the desired location and provide a file name. The format will be Ensight .case files.

4. When you select **Save**, the **Ensight Case File Options** dialog box opens.



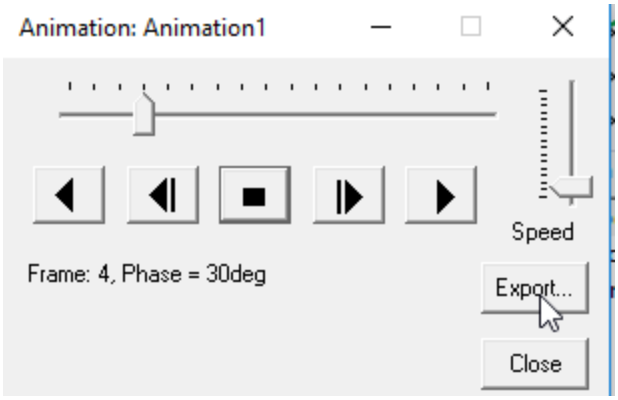
The binary option permits a more compact and efficient animation creation process in Ensight. The Single File options leverages the Ensight transient dataset capability to export all frames of field/geometry data to a single case file. On **OK**, every visible and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

animatable field plot should generate a case file with name in the format of “UserTypedName_fieldPlotName.case”

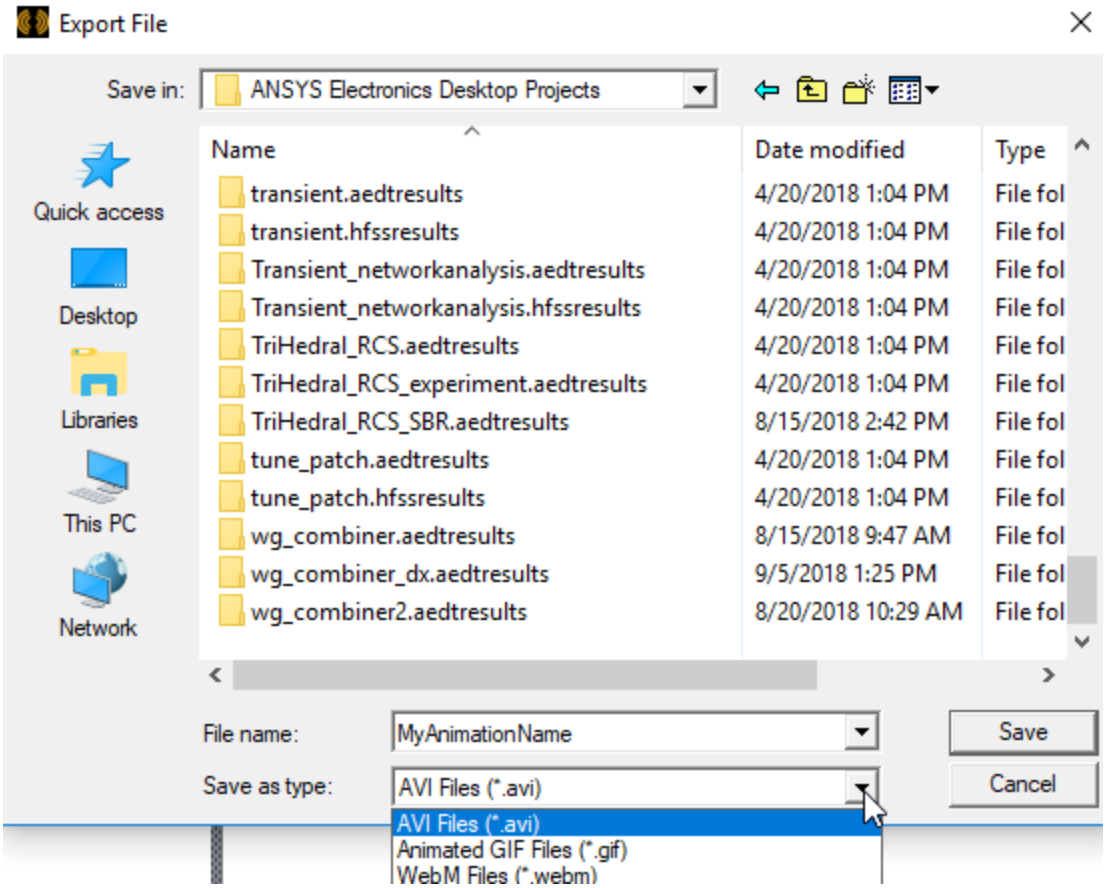
Exporting from the Animation Control Dialog Box

1. Create the animation you want to export.
2. In the play panel, click **Export...**.



The *Export File* dialog box appears.

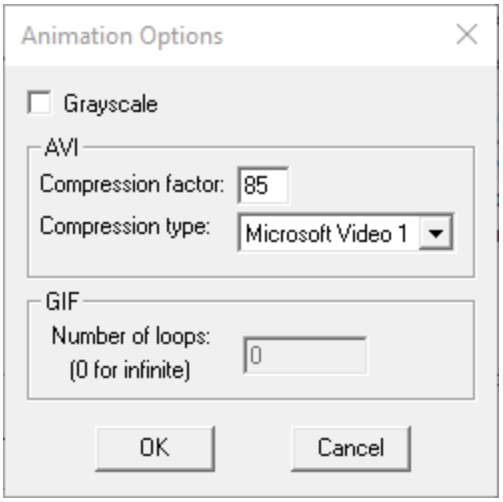
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

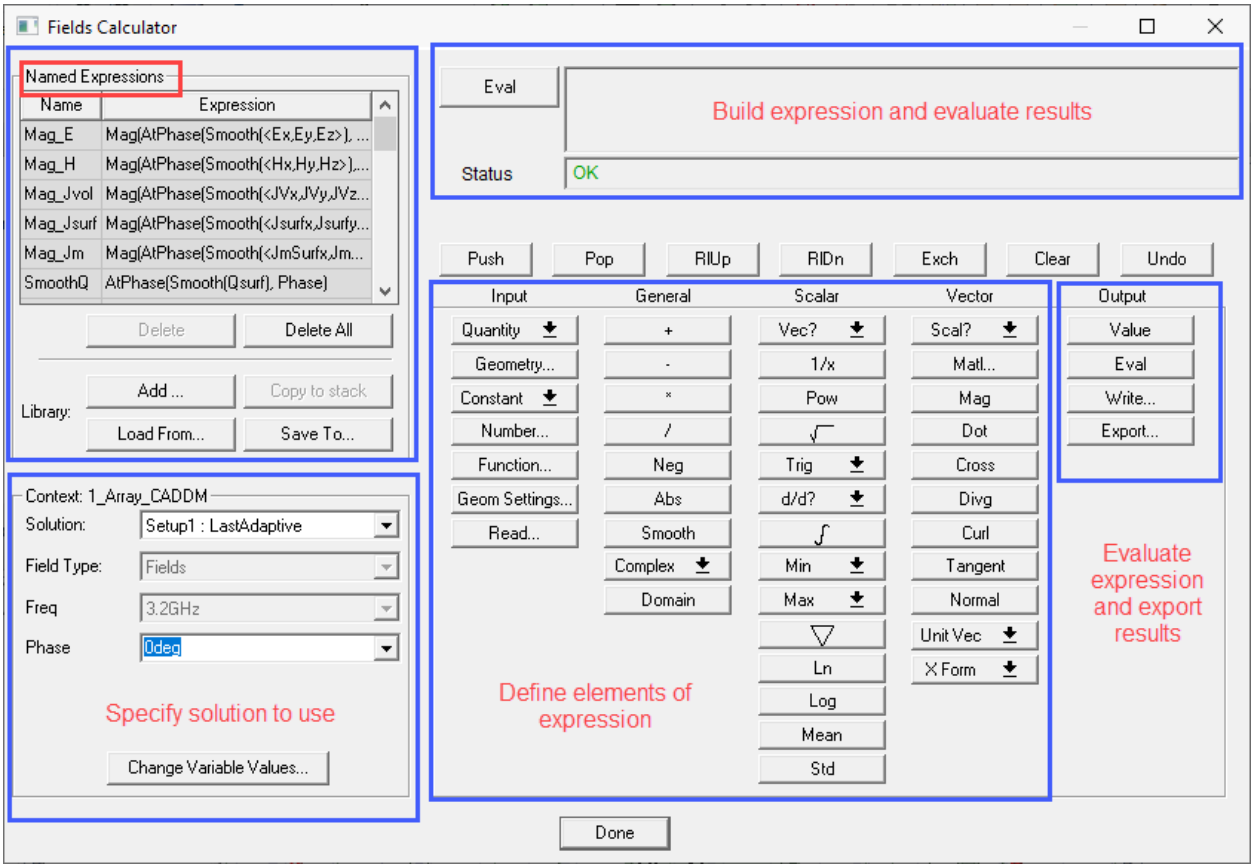
Using the Fields Calculator

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.

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 Calculator includes predefined expressions appropriate for each solver and lets you create and save additional named expressions.

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.

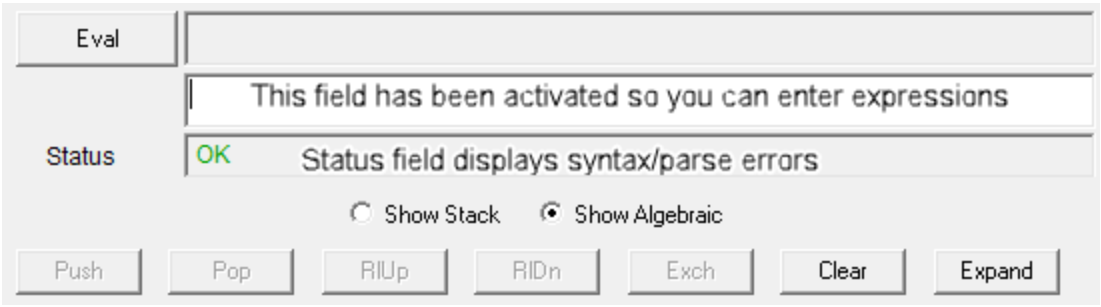
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The bottom 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.

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.

For HFSS 3D Layout, you can create custom field quantities using algebraic expressions. For more information on this functionality, see Algebraic Mode.



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:

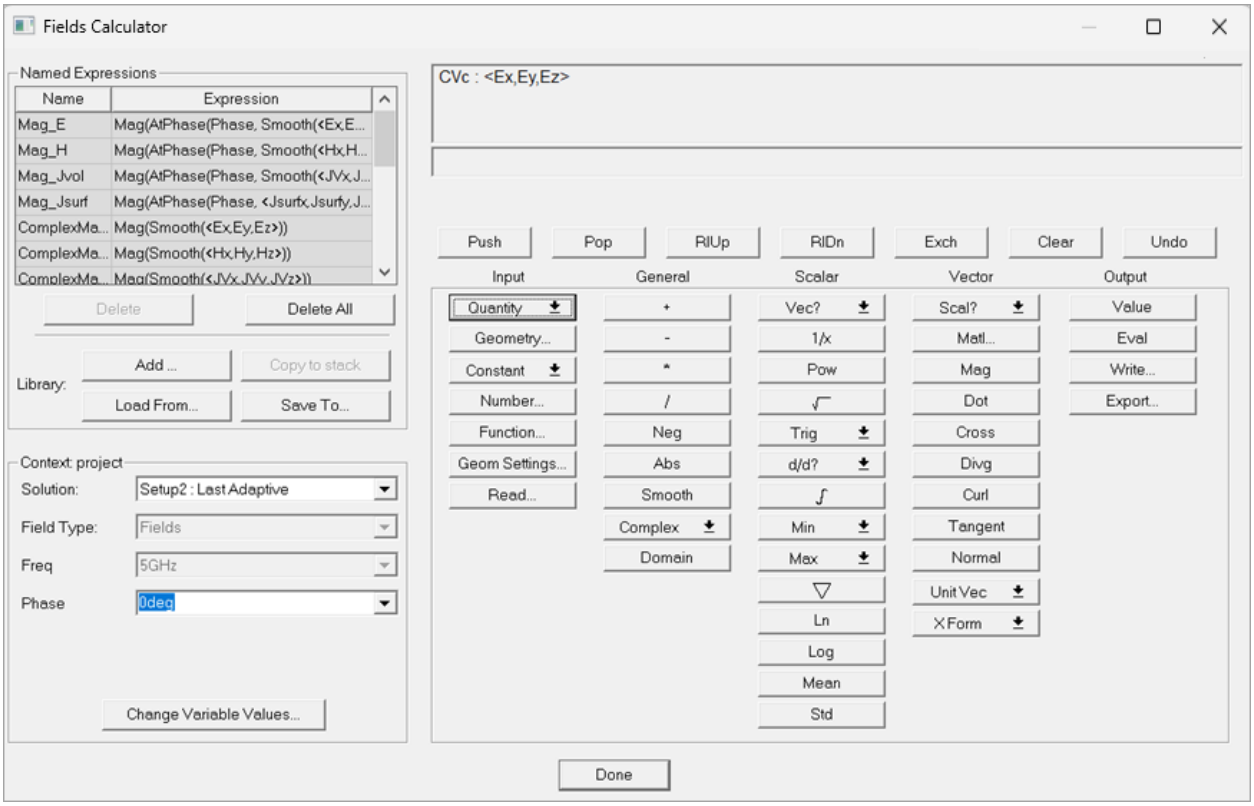
- Click **HFSS 3D Layout** > **Fields**> **Calculator**

  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.

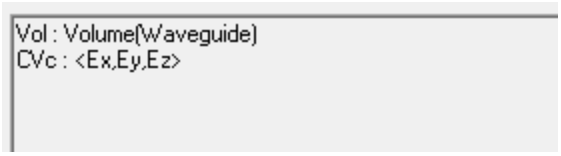
The Field Type here is not related to the edit sources. This is a general term among Ansys EM products (HFSS, Maxwell, Q3D, Icepak, and IcepakFEA). Some products have more than one field type for different solution types. If only one Field Type is available the drop-down menu is unavailable.

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.

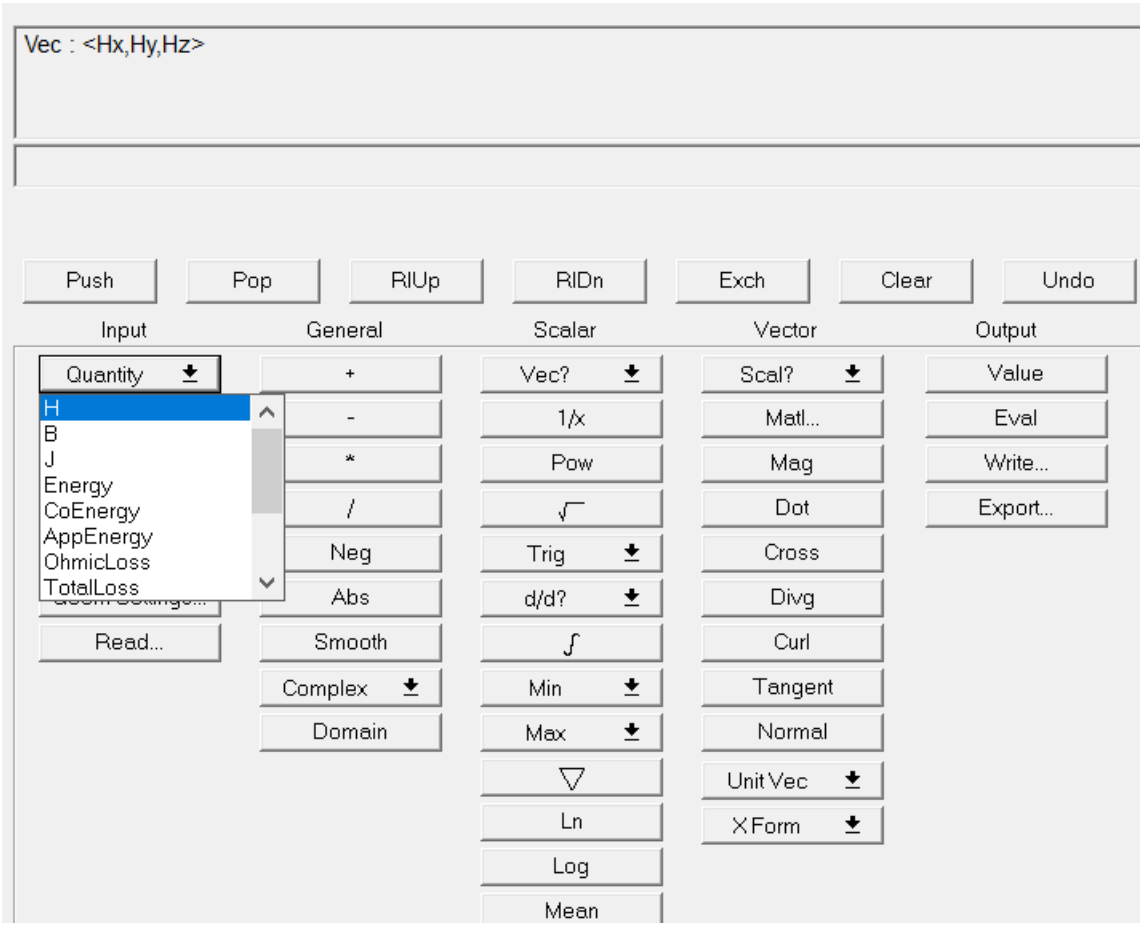
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:



Once a quantity is loaded into a register, it can be

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

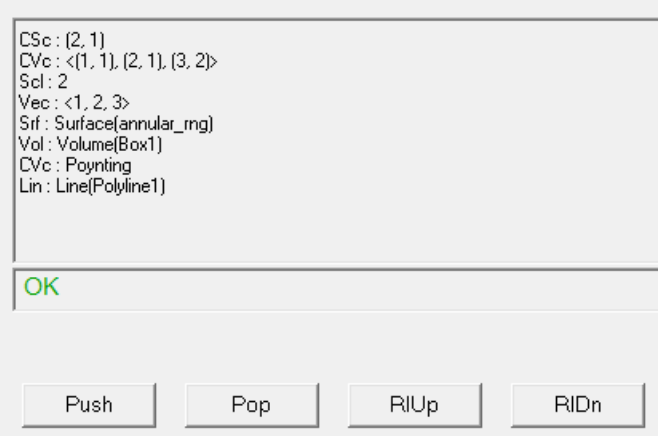
> **Note:**
>
> You cannot undo a simple operation such as loading a field quantity, constant, function, or geometry into the calculator. Instead, use the **Pop** or **Clear** commands to delete these items from the calculator stack.

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.

> **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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, 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.

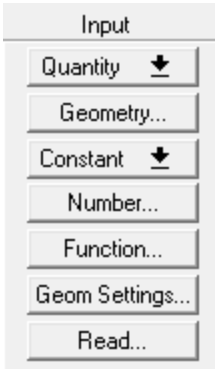
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Units of Measure

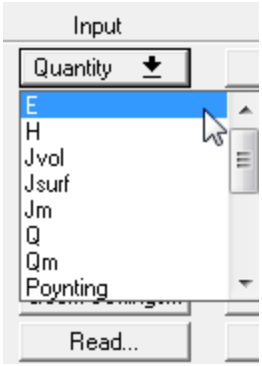
Unless you are prompted specifically for the unit of measure, all measurements should be assumed to be in SI base units, not model units.

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.



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.

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 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 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.

Temp	The resultant scalar temperature result, available from Thermal solutions.
Heat_Flux	A derived scalar quantity indicating the rate of heat flow on a per unit area basis. It is computed from the temperature field for Thermal solutions.
Surface_Loss_Density	This quantity concerns thermal excitations imported from the surfaces in another design. The electromagnetic losses are due to inherent inefficiencies in the materials, and these become a source of heat in the thermal solution.
Volume_Loss_Density	This quantity concerns thermal

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	excitations imported from the volumes in another design. The electromagnetic losses are due to inherent inefficiencies in the materials, and these become a source of heat in the thermal solution.
Displacement	The displacement vector results from modal solutions. The displacement direction and magnitude at one point on the model relative to another point demonstrates the shape of the vibration mode. However, the absolute magnitudes of the displacement are not meaningful because they are not scaled to any specific excitation. Rather, the displacement magnitudes are arbitrary for natural frequency (modal) solutions.

Geometry Command

The Fields Calculator **Geometry** command opens a dialog box for selecting a geometry to load into the top register of the calculator.

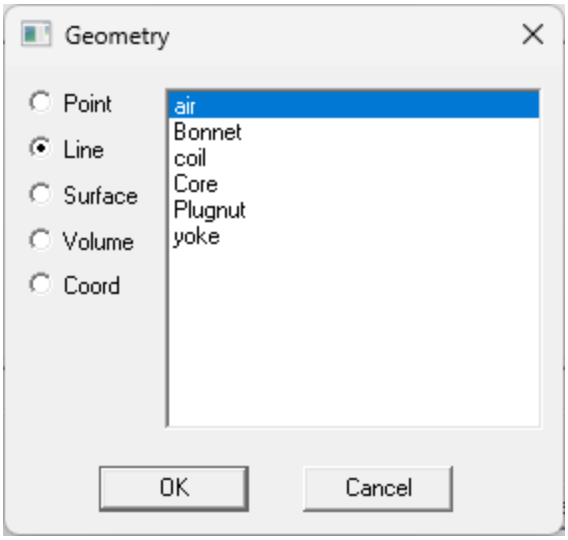
Do this to

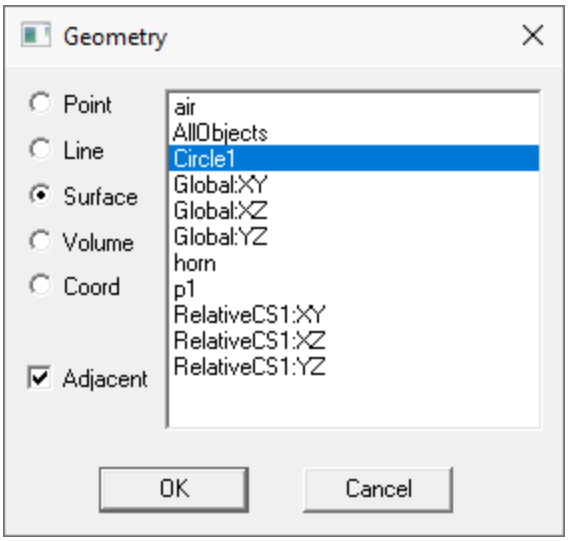
- find the value of derived field quantities on any point, line, surface, or volume.
- plot quantities directly from the calculator.
- display a previously defined isosurface, maximum or minimum field point using the **Draw** command.

The following types of geometries are available:

Point	See Drawing a Point Object. Drawn points are listed in the 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 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.





Surface

- Allow operation on the adjacent side surface geometry just like regular surface geometry.
- All calculation on this geometry will be done using data from the adjacent side of the surface.
- Support scripting, saving and loading expression with adjacent side surface geometry.

```
Srf : AdjacentSurface(Circle1)
CVc : <Ex,Ey,Ez>
```

Volume

3D objects, regions, and object selections (of 3D objects including AllObjects) are available in the calculator's *Geometry* 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 **Volume** is selected.
Coord	Coordinate systems are available in the calculator's *Geometry* dialog box when **Coord** is selected.

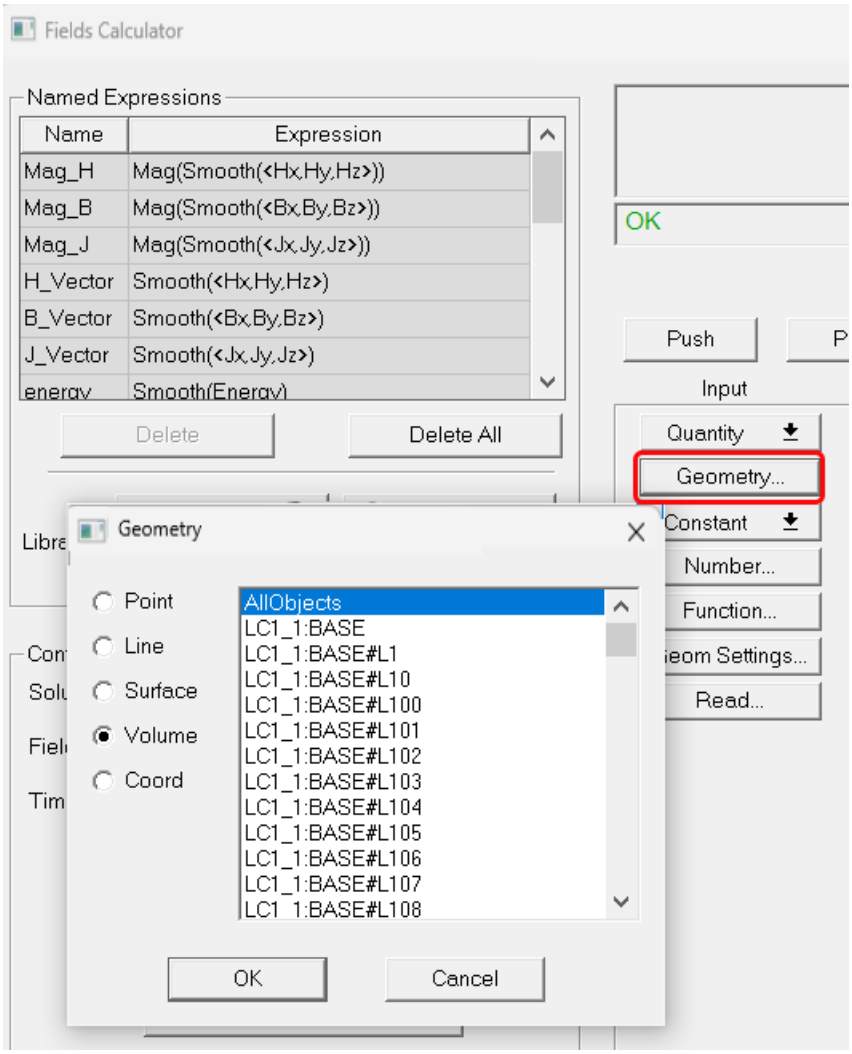
To load a geometry into the calculator:

1. In the Fields Calculator, click **Geometry**.

   The **Geometry** dialog box appears.

2. Select a geometry type.

   A list of all applicable geometries appears. If a design includes Layout Components, the Geometry list shows the available component geometries 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.

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 mesh, so they won't show up if there is no solution.

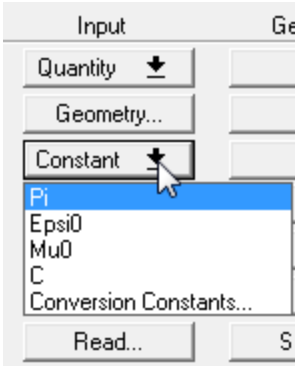
3. Click the geometry.
4. Click **OK** to load the geometry.

> **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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Pi π

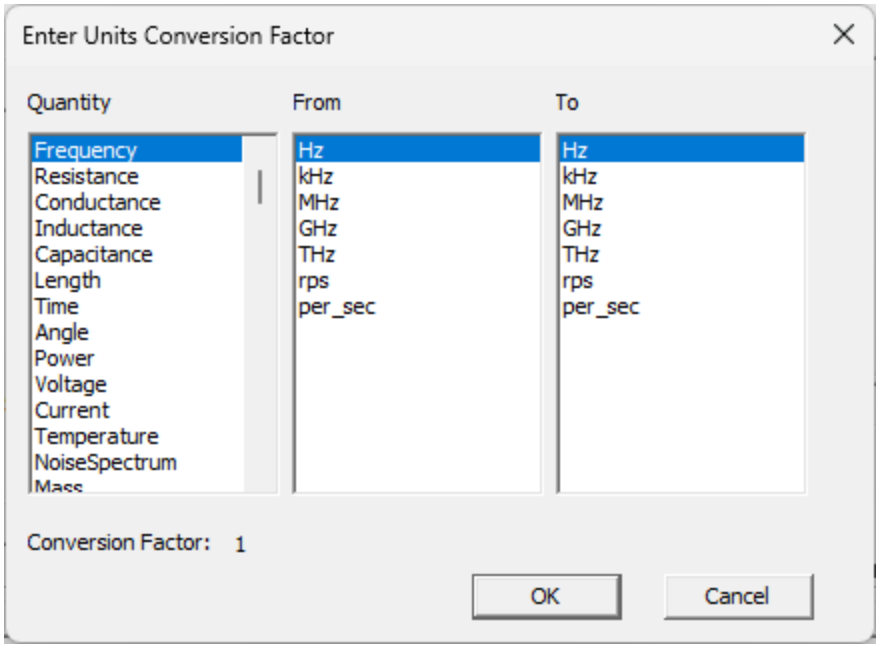
Epsi0 The permittivity of free space, $\varepsilon_0 = 8.85418782 \times 10^{-12}\ C^2/Nm^2$

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.



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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number Command

The **Number** command enters one of the following into the top register of the calculator:

Scalar	A scalar constant. To enter a constant scalar number: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Scalar**. 3. Type the scalar value in the **Value** text box. 4. Click **OK** to load the number into the top register.
Vector	A vector constant. To enter a constant vector: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Vector**. 3. Enter the x-, y-, and z-components of the vector. 4. Click **OK** to load the vector into the top register.
Complex	A complex constant, entered in the form $C=A+jB$, where A represents the real part of the constant and B represents the imaginary part. 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.

ad109dc8.png

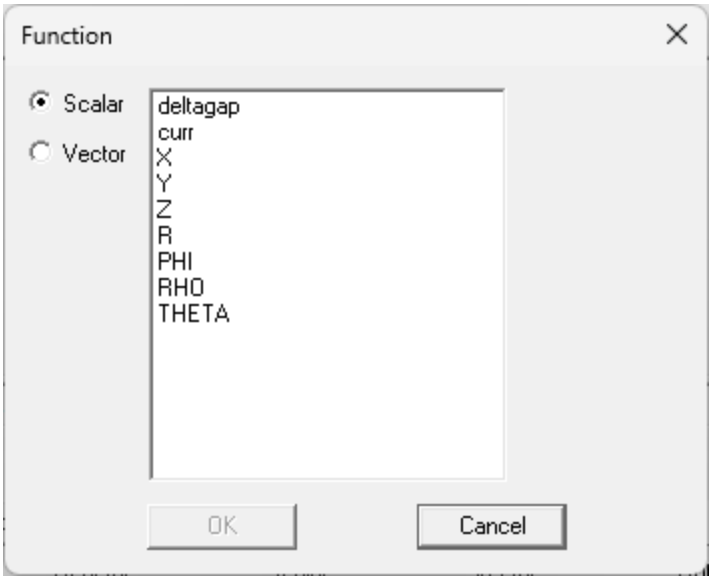
Function Command

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Function* dialog box appears.



2. Select **Scalar**.
3. Select the function from the list.
4. Click **OK** to load the functional scalar into the top register.

Vector

A vector function, in which the values of the vector's x-, y-, and z-components are given by functions.

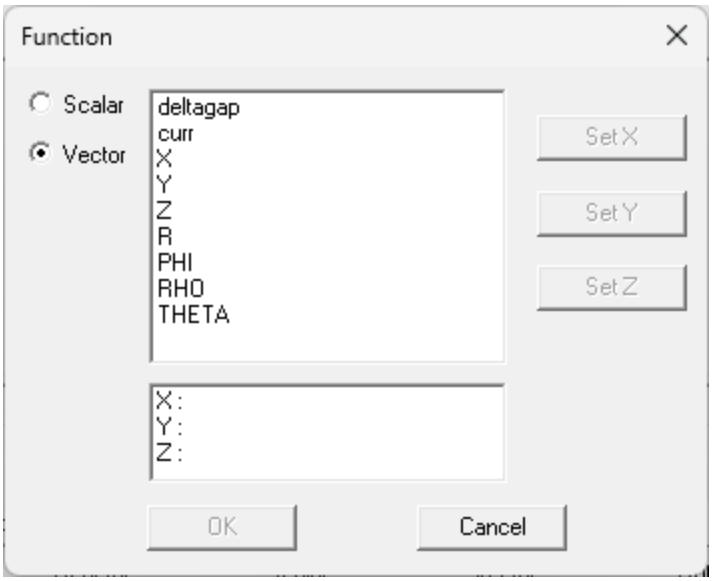
To enter a functional vector:

1. Click **Function**.

   The *Function* dialog box appears.

2. Select **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. Select the function from the list.
4. For each component of the vector, click **SetX, SetY,** and **SetZ**.
5. Click **OK** to load the functional vector into the top register.

Note:

The predefined variables **X, Y,** and **Z**; **RHO, THETA, R,** and **PHI**; and any functions that you created can be used to define functional scalar and vector quantities.

Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

Geom Settings Command

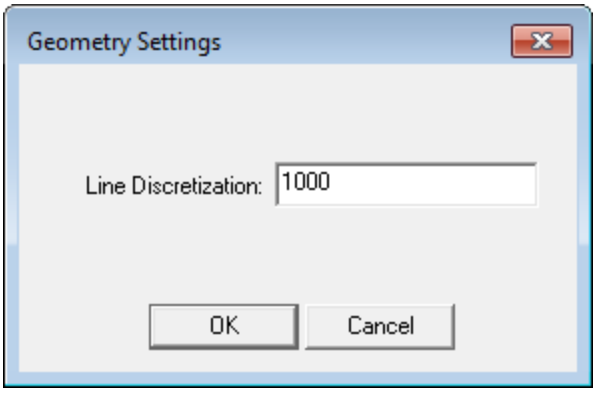
Clicking the **Geom Settings** button opens the *Geometry Settings* dialog box. This dialog box allows you to specify the line discretization, the number of equally spaced points used to integrate fields and other quantities on a line. The default is 1000 points.

To set the line spacing for geometry settings:

1. In the Fields Calculator, click **Geom Settings**.

   The *Geometry Settings* dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter a value in the **Line Discretization** text box and click **OK**.

Read Command

This **Read** command copies the contents of a disk file into the top register. The register must be one that has been saved using the Write output command.

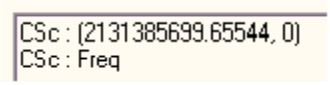
To read in a register:

1. Click **Read**.
2. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assumed for register files.
3. Click **OK**.

   The contents of the file are copied to the top register in the stack.

Output Vars [Input for Eigenmode problems]

This Output Vars button appears in the **Inputs** column of the Fields calculator only for Eigenmode problems. Freq is the only value listed. After you push Freq to the stack, you can click **Eval** to return a complex value.



General Commands

The General Column contains calculator operations which can be performed on many different data types (e.g., vector, scalar, complex). With the exception of the Complex menu, all are distinct functions. Most are self-explanatory, with the exception of Smooth which performs some data “smoothing” or statistical averaging on the top stack entry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use these Fields Calculator commands to perform operations on both vector and scalar quantities.

+ (Add)	/ (Divide)	Smooth
-- (Subtract)	Neg	Complex
* (Multiply)	Abs	Domain

+ (Add)

Adds the quantities in the top two registers of the calculator.

- (Subtract)

Subtracts the quantity in the top register from the quantity in the second register. The two registers must hold the same type of quantity (both scalar or both vector). You cannot subtract a scalar from a vector (or vice versa).

> **Note:** Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1.
>
> If you explicitly subtract Box2 from Box1, any calculation on the surface (faces) of Box1 will use the 6 exterior faces and the 6 interior faces. Any calculation on the volume of Box1 will use the difference in volume between Box1 and Box2.
>
> If you do not explicitly subtract Box2 from Box1, the inner box is only implicitly subtracted. Any calculation on the surface of Box1 in this case will use only the 6 exterior faces of Box1. Any calculation on the volume of Box1 will use the entire volume without subtracting the volume of Box2.

*** (Multiply)**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multiplies the quantity in the top register by the quantity in the second register. One of the two registers must contain a scalar value; the other register can be either a scalar or a vector.

/ (Divide)

Divides the quantity in the second register by the quantity in the top register. The second register must contain a scalar value; the top register can be either a scalar or a vector.

Neg

Changes the sign of the quantity in the top register.

Abs

Takes the absolute value of the quantity in the top register.

Smooth

Smooths the quantity in the top register. Because of the numerical solution technique used, field values are not always continuous across the boundaries of the individual elements that make up the finite-element mesh. Smoothing makes the values continuous by taking a weighted average from all of a node's neighboring elements. The weights are based on angles, so elements with larger angles provide larger contributions. In general, use smoothing before plotting a quantity.

Complex

These commands perform operations on a complex quantity in the top register. Complex quantities are indicated by a **C** at the beginning of the register label. They can be represented in terms of real and imaginary components, or in terms of magnitude and phase:

$$C = A + jB = Me^{j\phi}$$

where:

- *A* is the real part of the complex number.
- *B* is the imaginary part of the complex number.
- *M* is its magnitude, which is equal to $\sqrt{A^2 + B^2}$.
- ϕ is its phase, which is equal to **atan**(B/A).

The **Complex** commands includes a drop-down menu that let you do the following:

Real	Takes the real part of the complex quantity (A).
Imag	Takes the imaginary part of the complex quantity (B).
CmplxMag	Takes the magnitude of the complex quantity (M). Due to interpolation issues, the sequence of calculations may cause a loss of accuracy. It is best to define the points , separately obtain the value of the real part, then the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	imaginary part, and use those values to calculate the magnitude and and phase. For the sequence for using the Fields Calculator to obtain the real and imaginary parts, see the procedure here.
CmplxPhase	Takes the phase of the complex quantity(ϕ).
Conj	Takes the complex conjugate of the quantity in the top register. If a complex number is given by $C = A + jB$, its complex conjugate is given by $C^* = A - jB$.
AtPhase	Lets you specify the phase angle, θ, at which an field quantity is evaluated. These quantities can be represented in the form $A(x,y,z,t) = A(x,y,z)\cos(\omega t + \theta(x,y,z))$. where • ω is the angular frequency at which the quantities are oscillating, specified during the solution. • θ(x,y,z) is the phase angle (the offset from a cosine wave that peaks at t=0). Entering the phase angle lets you compute the real part of the field's magnitude at different points in its cycle.
CmplxReal	Converts the real scalar of the top register to the real part of a complex number.
CmplxImag	Converts the real scalar of the top register to the imaginary part of a complex number.
CmplxPeak	Calculates the peak value of a given complex vector. Intuitively, this calculates the maximum magnitude of the equivalent real vector in a waveform.

Domain

This limits a calculation to the volume you specify. The domain filter works for scalars, vectors, complex scalars and complex vectors. This operation requires the top two entries of the stack to be a volume geometry and a numeric field quantity. To do this:

1. Load the field quantity into the top register, and perform any necessary operations on it.
2. Load the volume using the **Geometry** command.
3. Click **Domain**.

The **Domain** command is often used to limit a calculation or plot to the intersection of a surface and an object or group of objects. If you export a domain filtered numeric, points that are filtered out by the domain will not be written out.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 HFSS 3D Layout 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.

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 top register value is 1.5, and the second register value is 2, the ATan2 function returns ATan(2/1.5) = 0.5880 radians. All other Trig functions operate on a single argument (top register only).
d/d?	Takes the partial derivative of the quantity in the top register.
∫? (Integral)	Takes the integral of a scalar quantity over a volume, surface, or line. To perform integration upon complex quantities, you must perform the integration separately on real and imaginary subcomponents.
Min	Computes the minimum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
Max	Computes the maximum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
∇ (Gradient)	Takes the gradient of the scalar quantity in the top register.
ln	Takes the natural logarithm (base e) of the scalar quantity in the top register.
log	Takes the logarithm (base 10) of the scalar quantity in the top register.
Mean	Takes the mean of the quantity in the top register.
Std	Takes the standard deviation of the quantity in the top register.

Vec? Command

Makes the scalar quantity in the top register a vector component. Choose from the following:

VecX	The x-component of a vector.
VecY	The y-component of a vector.
VecZ	The z-component of a vector.

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.

Trigonometry Commands

Takes one of the following trigonometric values of the value in the top register of the calculator stack:

Sin	Sine
Cos	Cosine
Tan	Tangent
Asin	Arcsine
Acos	Arccosine
Atan	Arctangent
Atan2	A two-argument version of the Atan function. Takes the y and x coordinates of the tangent point (in the top two stack registers)

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.

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 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. HFSS 3D Layout integrates the tangential or normal component of the quantity over the line on which you computed its tangent, or the surface on which you computed its normal.

3. Choose the ∫ command to integrate the scalar quantity over the geometry.

To find the numerical results of an integration, use the **Eval** command.

Min Command

Computes the minimum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the minimum value of the field.
Position	Finds the point where the minimum field value occurs. You can then: • Plot the minimum field value at the point. • Plot basic field quantities at the point. • Load the point into the calculator. • Change the point's location.

These commands operate in the same way as the **Max** commands. Use the **Eval** command to display the actual minimum field value or the coordinates of the point where it occurs.

Max Command

Computes the maximum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the maximum value of the field.
Position	Finds the point where the maximum field value occurs. You can then: • Plot the maximum field at the point.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

Gradient Command

Takes the gradient (∇) of the scalar quantity in the top register.

Ln Command

Takes the natural logarithm (base e) of the scalar quantity in the top register.

Log Command

Takes the logarithm (base 10) of the scalar quantity in the top register.

Takes the mean of the quantity in the top register.

Std Command

Takes the standard deviation of the quantity in the top register.

Vector Commands

The Vector Column contains calculator operations which can only be performed on vector stack entries. Drop-down menus in this column are Scal? (convert vector to scalar) and Unit Vec (create unit vector). Standard vector algebra operations (Dot, Cross, etc.) are also present.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use these commands to perform operations on vector quantities.

Scal?	Replaces the vector in the top register with a scalar quantity whose value is a component of the vector, your choice of ScalarX, Scalar Y, or ScalarZ.
Matl	Multiplies or divides the vector field quantity in the top register by a material property, or if you select MassDensity as the material property, produces a scalar that operates like a named variable.
Mag	Takes the magnitude of the vector quantity in the top register. The magnitude of a complex vector is defined to be the length of the real vector resulting from taking the modulus of each component of the original complex vector.
Dot	Takes the dot product of the vector quantities in the top two registers.
Cross	Takes the cross product of the vector quantities in the top two registers.
Divg	Takes the divergence of the vector quantity in the top register.
Curl	Takes the curl of the vector quantity in the top register.
Tangent	Computes the tangential component of a vector quantity along a line
Normal	Computes the normal component of a vector quantity on a surface such as a cutplane or object surface.
Unit Vec	Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation. The drop-down lets you select from Tangent, Normal, or CoordSys[X, or Y, or Z]
X Form	Transforms to one of the following: 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.

Scal? Command

Replaces the vector in the top register with a scalar quantity whose value is a component of the vector. Choose from the following:

ScalarX	Returns the x-component of the vector.
ScalarY	Returns the y-component of the vector.
ScalarZ	Returns the z-component of the vector.

Matl Command

Operates on the vector field quantity in the top register of the Fields Calculator based on a material property. At each tetrahedron, the field quantity is operated on by the value of the selected material property — taking the different material attributes of each object into account.

To operate on a vector quantity by a material property:

1. Click **Matl**.

   The *Material Operation* window appears.

2. Select a material property. Available properties are

Permittivity (epsi)	The permittivity, ε.
Permeability (mu)	The permeability, μ.
Conductivity	The conductivity, σ.
Omega (w)	The angular frequency, ω. The angular frequency is equal to $2\pi f$, where f is the frequency at which the solution was generated.
MassDensity	This is based on the value of the Mass Density material property. MassDensity is treated like a named expression. Selecting MassDensity disables the Operation radio buttons for Multiply or Divide in the *Material Operations* dialog box.

> **Warning:**
>
> The Matl command is not compatible with nonlinear 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.

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

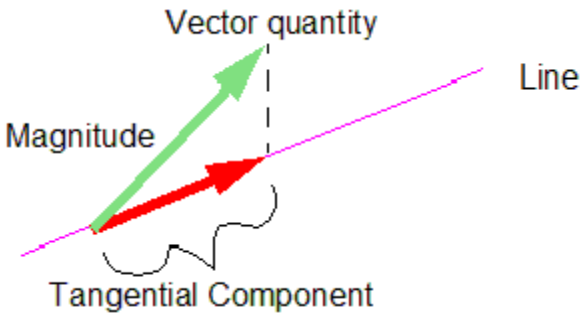
Takes the divergence of the vector quantity in the top register.

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.

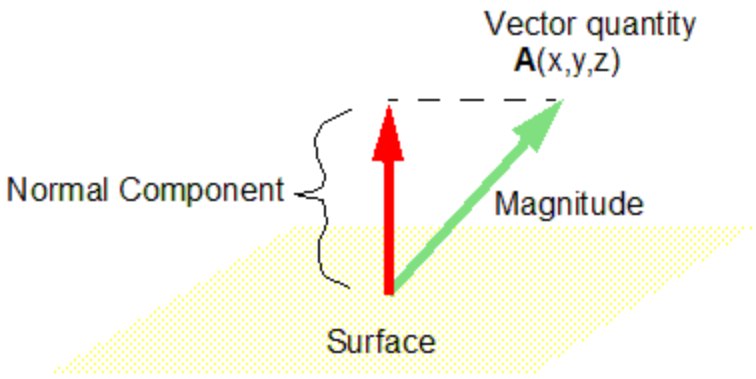
To take the tangent of a vector:

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**.

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 HFSS 3D Layout, the fields calculator uses the surface normal of the face of the 3D object that is coincident with the sheet.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit Vec Command

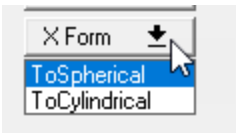
Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation.

Select from the following:

Tangent	Computes the unit vector tangent to the line specified at the time of plotting, integrating, or report generation based on the context.
Normal	Computes the unit vector normal to the surface specified at the time of plotting, integrating, or report generation based on the context.
CoordSys (X)	Computes the unit vector in the X-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Y)	Computes the unit vector in the Y-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Z)	Computes the unit vector in the Z-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.

X Form

Fields are defined in the Cartesian coordinate system. In many applications it is much more convenient to use a cylindrical or spherical coordinate system – for example, when the object is of cylindrical or spherical shape. The **XForm** command allows the user to transform vectors into either the cylindrical or spherical coordinate system. Selecting **XForm** displays a selection menu for the coordinate system (**ToCylindrical** or **ToSpherical**):

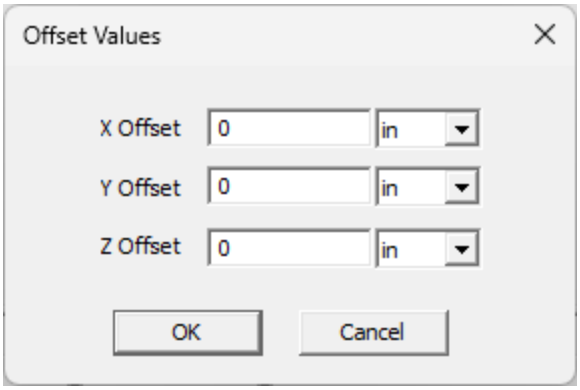


ToSpherical – Transforms the vector in the top register of the calculator stack into the spherical coordinate system.

ToCylindrical – Transforms the vector in the top register of the calculator stack into the cylindrical coordinate system.

After you select the coordinate system you want, the **Offset Values** dialog displays.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you enter X, Y, and Z offsets to be used as the origin of the new coordinate system.

Click **OK** to compute the offset coordinate transformation to apply to the (complex) vector quantity in the top register on the calculator stack based on your selection of Cylindrical or Spherical coordinates and your input of X, Y, and Z offsets. The new expression will be pushed onto the stack. This provides the origin of the new coordinate system. It will be something like

CVc : ToCylindrical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

or

CVc : ToSpherical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

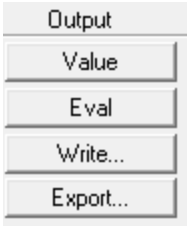
After you transform the quantities, you can then access the vector components in the Fields Calculator.

You will then use the ScalarX/ScalarY/ScalarZ commands under Scal?. For a vector converted to cylindrical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component, and ScalarZ gives the Z component. For a vector in spherical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component and ScalarZ gives the theta (azimuthal) component.

Output Commands

The Output Column contains those calculator operations that result in final data outputs from calculations. The Eval button obtains final numerical results from the last stack placeholder (such as integrations).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

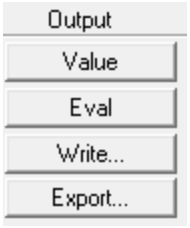


Use these commands to compute or evaluate expressions and to output the data in the calculator.

Value command	Computes the value of a field quantity on a geometry.
Eval command	Numerically evaluates and displays the results of calculator operations from the last stack placeholder, such as integrations.
Write command	Saves the contents of the top register to a disk file for future re-use.
Export command	Saves field quantities in a format that can be read by other modeling or post-processing software packages.

To plot/tabulate the extracted data using create report or create field overlays, you should use a named expression.

Value Command



This computes the value of a field quantity at a point. Use it to find:

- The magnitude of a scalar field quantity at that point.
- The x-, y-, and z-components of a vector field quantity at that point.

The **Value** command applies the value of the next-to-top stack entry (a field quantity) on the geometry entry at the top stack entry. The field quantity entry may be scalars, real or complex vectors. The geometry entry may be of any type (point, surface, line, or volume).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, if the top stack entry in the Field Calculator is a surface and the second stack register is for the real part of the E Vector ("Real(<Ex, Ey, Ez>)"), the resulting quantity will be an expression for the real part of the E vector on the surface (a VecSrf quantity). Again, the result of this command is still a placeholder expression, not the final numerical values.

Strictly speaking, the Value command provides access to the field quantity, not data outputting. It simply provides a handle to the (numerical) values of a field quantity on a given geometry, and thus the outcome of the command is not numerical values themselves. If the numerical value of a field quantity is desired, you will need to follow it up with one of the three other output commands, or define a named expression for the quantity and plot it with the Reporter.

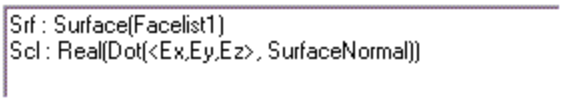
To find the value of a field quantity at a point:

1. Load the field quantity into the top register, and perform any needed operations on it.
2. Load the appropriate point into the calculator using the **Geometry/Point** command.
3. Click **Value**.

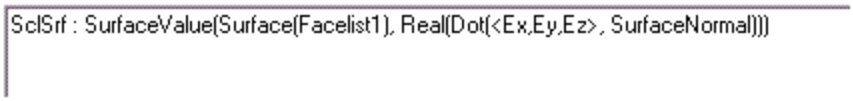
To view the numerical results of this operation, use the **Eval** command.

You can also use the **Value** command to access the intermediate SurfaceValue function.

For example, after inputting an expression for a quantity, such as an E field, and then selecting a surface geometry, the calculator stack displays something like this:



Clicking the **Value** command changes the display to the following, showing the intermediate SurfaceValue function.



In this case, SurfaceValue provides the x, y, z, coordinates of the FEM mesh and Lagrangian points so you can use **Write** to generate an .fld file containing an evaluated scalar quantity at those points.

In general for **Value**:

1. Enter any quantity onto the stack.
2. Enter a volume / surface / line / point onto the stack.
3. Press the **Value** button and you will get an appropriate geometry value on the stack.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

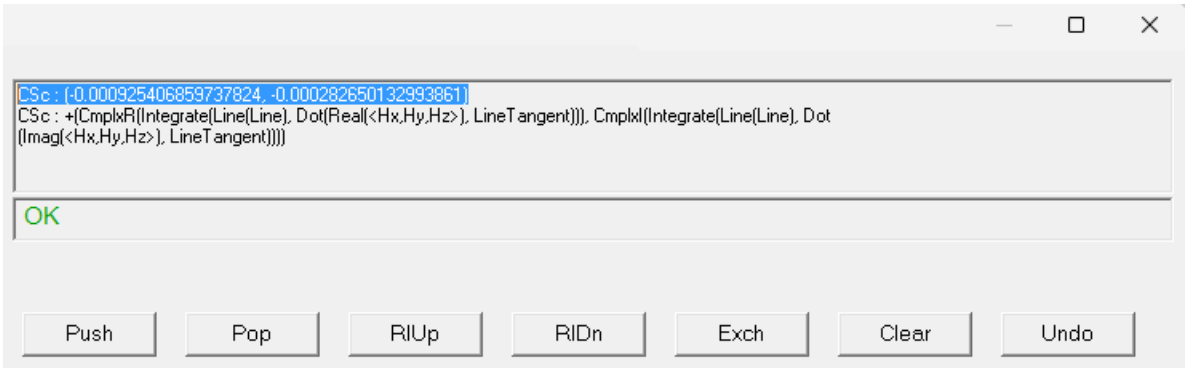
Now you can perform suitable operations such as Write, Integrate, etc. For PointValue you can also do **Eval**.

Eval Command

The **Eval** command finalizes computations and converts the text-string indicating the computation being performed to an actual numerical value. This command numerically evaluates and displays the results of calculator operations such as integrations, maximum or minimum field computations, field values at points, and so forth. For this command to work properly, the stack entry must be reduced to a single-value entry which can be a scalar, complex, or vector (real or complex). The quantity to be evaluated must be in the top register. The **Eval** command computes the numerical results of the operation, which replace the contents of the register. Units of the numerical value are not provided.

For a calculator expression that evaluates to a single value, the **Eval** command offers a quicker way to obtaining the final numerical result than defining a named expression and then trying to plot/tabulate it in the reporter. But for many expressions that don't evaluate to single values, such as the wave impedance along a line mentioned earlier, **Eval** command does not apply. As such, use of named expression and the Report Editor will be the only option.

Evaluate the expression for voltage along a line shows the definition of an expression for the voltage between two conductors by integrating the electric field along a line using the recipe given for Calculating the Current along a Wire or Trace. Then, by clicking the Evaluate command in the Output column, we obtain the numerical value for the voltage as shown in the top of the stack. It can then be either written down or copied/pasted to other programs.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

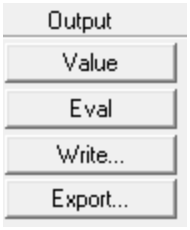
$$I = \oint H \cdot dl$$

Because **H** and **I** are complex quantities, you will need to evaluate the real part of **H** to obtain the real part of **I**, then evaluate the imaginary part of **H** to obtain the imaginary part of **I**. To do this:

1. Load **H** into the calculator using the **Qty** command.
2. Take the real part of **H** using the **Cmplx/Real** command.
3. Load the rectangular loop using the **Geom/Line** command. Create the loop, a closed polyline, to integrate over.
4. Click **Tangent** to get the component of **H** along the line.
5. Take the integral around the loop using the ∫ command.
6. Click **Eval** to evaluate the integral. The real part of **I** appears in the top register.
7. Repeat this process using the imaginary part of **H** (found with the **Cmplx/Imag** command) to obtain the imaginary part of **I**.

Write Command

If the calculator operations performed have obtained a stack entry that is intended for use in still other calculator operations, the stack entry can be saved for this purpose by using the **Write** button in the calculator Output column.



Note that in this case no name will be requested for the expression; only a save filename location will be required. This function will not work for field values derived upon a specific geometric quantity (those containing either Lin, Srf, or Vol in the stack data type indicator) as the calculator cannot know that these geometric quantities exist in identical forms in other postprocessing sessions.

When you try to read a file written with the **Write** command by clicking the **Read** button in the Input column, you must ensure that the mesh used for generating the current solution is the same as the one with which the saved field data is generated. Since the two results must be generated on the same mesh, not only the geometries of the designs on which the **Write/Read** commands were applied must be the same, they must also have gone through the same adaptive meshing process. In practice, this limits the use of Write/Read command to solutions for different frequency sweeps of the same analysis (adaptive) setup of the same design.

This command saves the contents of the top register to a disk file. Use this command to:

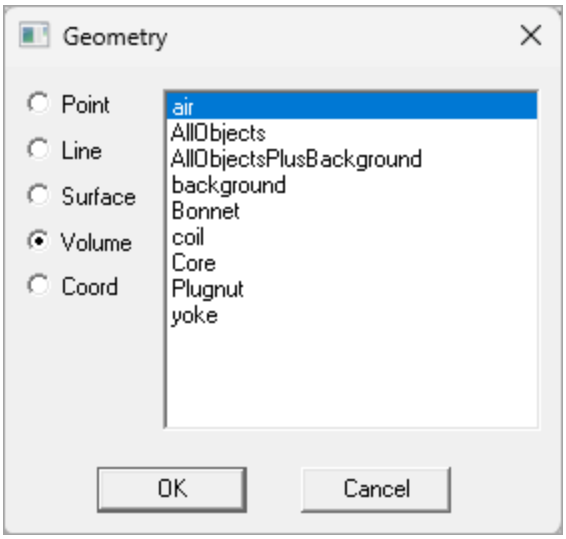
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Save registers for use during a later postprocessing session.
- Save a field quantity for use when postprocessing a different model.

> **Note:**
>
> Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

To save a register:

1. Click **Write**.
2. If the register includes numeric data with a constrained quantity (such as jsurf), you see a dialog box that gives a choice of constraining geometries. For example:



3. Select the geometry of interest, and select **OK**.

   This displays a file browser.

4. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assigned to register files and a .fld extension is assigned to field files. You can choose to save both .reg and .fld files, or either one.
5. Click **OK**.

   The contents of the register are saved to the file you specified.

Export Command

To output a field quantity or calculation result for use by some third-party post-processors, use the **Export** button in the calculator Output column. You can map the field quantity to either a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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:

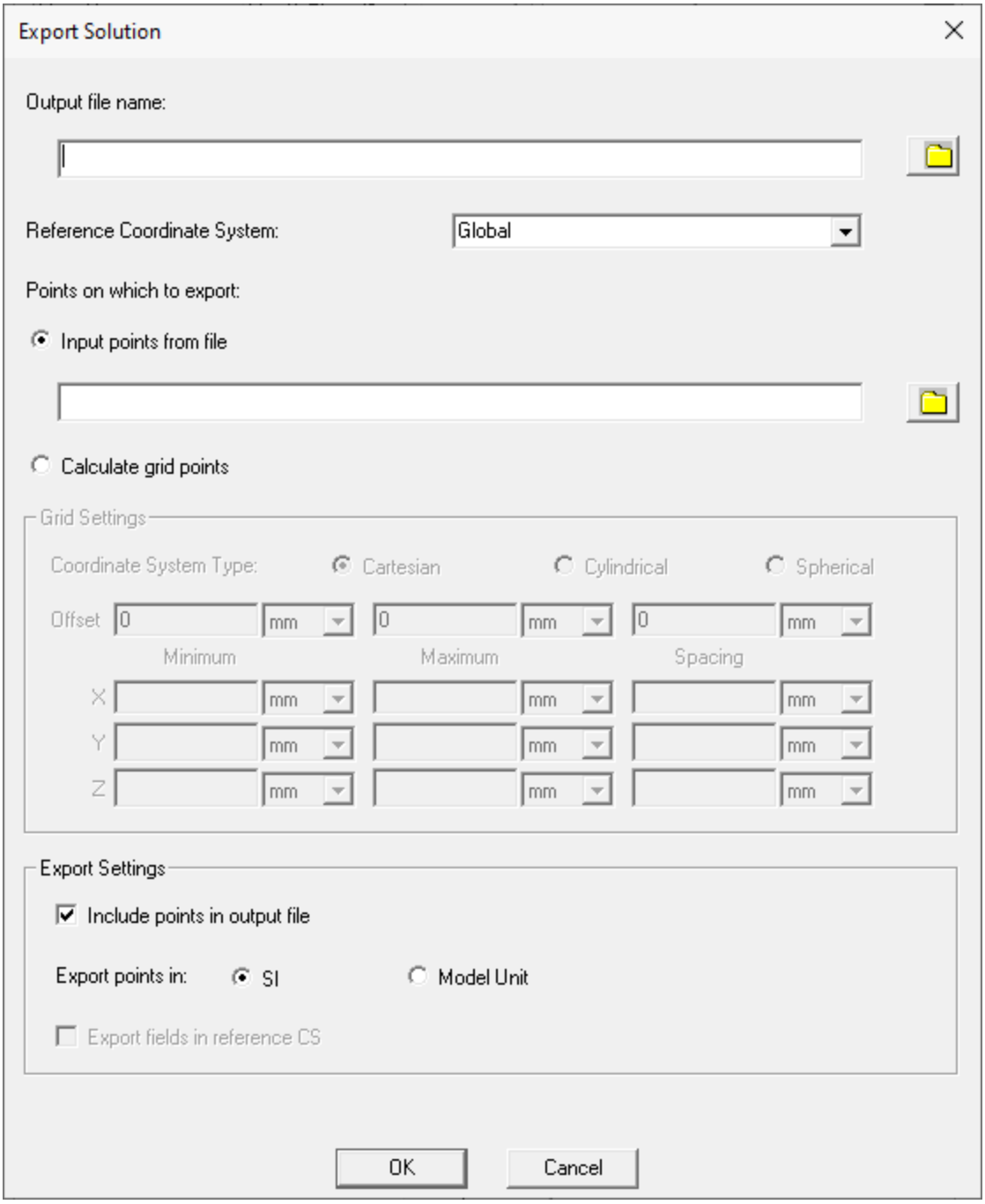
1. Load the quantity into the top register for the fields calculator, and perform any operations on it.
2. If desired, load a volume using the **Geometry** command.

   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.

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 **Export** button in the Fields Calculator.

   This opens the **Export Solution** dialog box.



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.

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 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 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

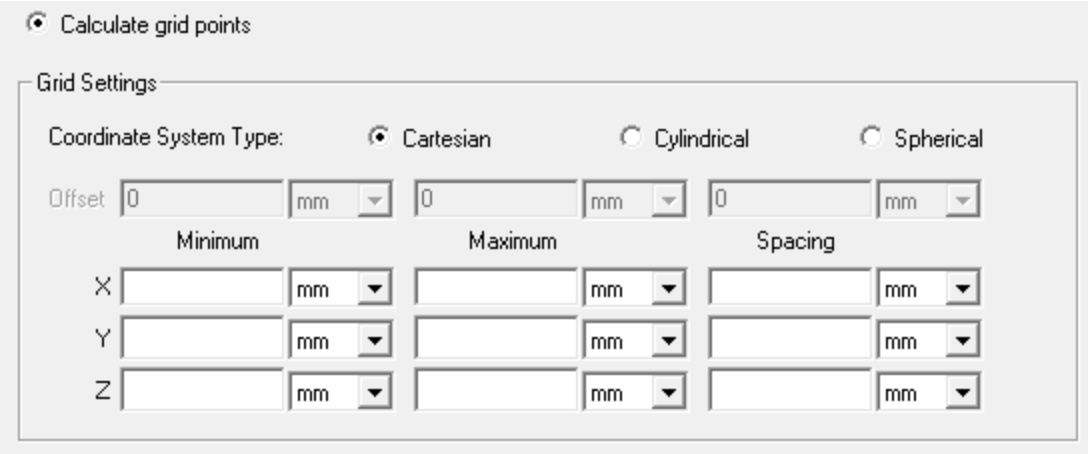
     -5.5 -5.5 -5.07375

     -5.5 -5.5 -5.021

   - If you select **Calculate grid points** button, you can specify the coordinate system

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as Cartesian, Cylindrical, or Spherical.



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.

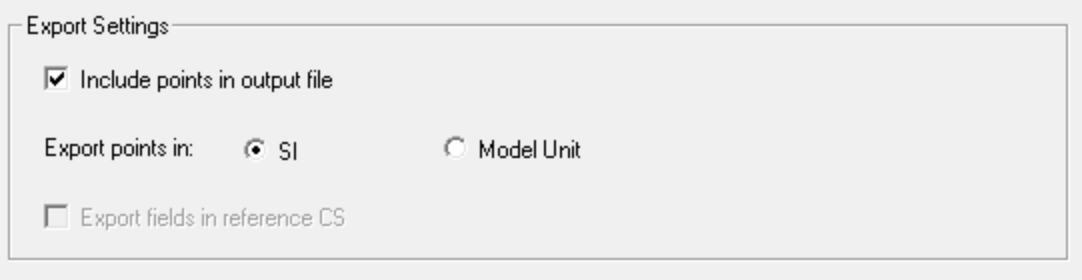
Spherical: for each dimension R, Theta, and Phi, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

> **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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).

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 HFSS 3D Layout design is open and a Solution Setup has been performed, the following predefined named expressions are available.

> **Note:**
>
> For a list of predefined named expressions for Q3D solution setups, refer to List of Field Quantities in a Q3D 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.

Expression Name	Expression Definition
Abs_Q	CmplxMag(Smooth(Q))
ComplexMag_E	Mag(Smooth(<Ex,Ey,Ez>))
ComplexMag_H	Mag(Smooth(<Hx,Hy,Hz>))
ComplexMag_ JdcThinCond	Mag(Smooth (<dcfSurfJx,dcfSurfJy,dcfSurfJz>))
ComplexMag_ SurfaceJac	Mag(Smooth(<ACRL_SurfaceJacx,ACRL_ SurfaceJacy,ACRL_SurfaceJacz>))
ComplexMag_ SurfaceJdc	Mag(Smooth(<Jx,Jy,Jz>))
ComplexMag_ VolumeJdc	Mag(Smooth(<dcvJx,dcvJy,dcvJz>))
Harmonic_Loss_ Density	Smooth(HarmonicLoss)
JdcThinCond	AtPhase(Phase, Smooth (<dcfSurfJx,dcfSurfJy,dcfSurfJz>))
Mag_E	Mag(AtPhase(Phase, Smooth(<Ex,Ey,Ez>)))
Mag_H	Mag(AtPhase(Phase, Smooth (<Hx,Hy,Hz>)))
Mag_JdcThinCond	Mag(AtPhase(Phase, Smooth (<dcfSurfJx,dcfSurfJy,dcfSurfJz>)))
Mag_SurfaceJac	Mag(AtPhase(Phase, Smooth (<SurfaceJacx,SurfaceJacy,SurfaceJacz>)))
Mag_SurfaceJdc	Mag(AtPhase(Phase, Smooth(<Jx,Jy,Jz>)))
Mag_VolumeJdc	Mag(AtPhase(Phase, Smooth (<dcvJx,dcvyJ,dcvJz>)))
Phidc	AtPhase(Phase, Smooth(dcvPhi))
PhidcThinCond	AtPhase(Phase, Smooth(dcfPhi))
SmoothQ	AtPhase(Phase, Smooth(Q))
Surface_Loss_ DensityAc	Smooth(SurfaceLossDensityAc)
SurfaceJac	AtPhase(Phase,Smooth (<SurfaceJacx,SurfaceJacy,SurfaceJacz>))
SurfaceJdc	AtPhase(Phase, Smooth(<Jx,Jy,Jz>))
Temperature_Ac_ Surface	Smooth(TemperatureAcSurf)
Temperature_Dc_ Volume	Smooth(TemperatureDcVol)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vector_H	AtPhase(Phase, Smooth(<Hx,Hy,Hz>))
Vector_E	AtPhase(Phase, Smooth(<Ex,Ey,Ez>))
Volume_Loss_ DensityDc	Smooth(dcvLossDensity)
VolumeJdc	AtPhase(Phase, Smooth (<dcvJz,dcvJy,dcvJz>))

List of Field Quantities in a Q3D Solution Setup

The following field quantities are available in Q3D Extractor, depending on the solution.

DC R/L Fields

VolumeJdc	AtPhase(Smooth (<dcvJz,dcvJy,dcvJz>),Phase)
Mag_VolumeJdc	Mag(AtPhase(Smooth(<dcvJx,dcvyJ,dcvJz>), Phase))
CmplxMag_VolumeJdc	Mag(Smooth(<dcvJx,dcvJy,dcvJz>))
Phidc	AtPhase(Smooth(DCRL_dcvPhi), Phase)
Volume_Loss_DensityDc	Smooth(DCRL_dcvLossDensity)

AC R/L Fields

SurfaceJac	AtPhase(Smooth(<ACRL_ SurfaceJacx,ACRL_SurfaceJacy,ACRL_ SurfaceJacz>), Phase)
Mag_SurfaceJac	Mag(AtPhase(Smooth(<ACRL_ SurfaceJacx,ACRL_SurfaceJacy,ACRL_ SurfaceJacz>), Phase))
CmplxMag_SurfaceJac	Mag(Smooth(<ACRL_SurfaceJacx,ACRL_ SurfaceJacy,ACRL_SurfaceJacz>))
Vector_H	AtPhase(Smooth(<ACRL_Hx,ACRL_ Hy,ACRL_Hz>), Phase)
Mag_H	Mag(AtPhase(Phase, Smooth(<ACRL_ Hx,ACRL_Hy,ACRL_Hz>), Phase))
CmplxMag_H	Mag(Smooth(<ACRL_Hx,ACRL_Hy,ACRL_ Hz>))

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CG Fields

SmoothQ	AtPhase(Smooth(CG_Q), Phase)
ABS_Q	CmplxMag(Smooth(CG_Q))
Mag_E	Mag(AtPhase(Smooth(<CG_Ex,CG_Ey,CG_Ez>), Phase))
ComplexMag_E	Mag(Smooth(<CG_Ex,CG_Ey,CG_Ez>))
Vector_E	AtPhase(Smooth(<CG_Ex,CG_Ey,CG_Ez>), Phase)

DC R/L Thin Conductor Fields

JdcThinCond	AtPhase(Smooth (<dcfSurfJx,dcfSurfJy,dcfSurfJz>), Phase)
Mag_JdcThinCond	Mag(AtPhase(Smooth (<dcfSurfJx,dcfSurfJy,dcfSurfJz>), Phase))
CmplxMag_JdcThinCond	Mag(Smooth (<dcfSurfJx,dcfSurfJy,dcfSurfJz>))
PhidcThinCond	AtPhase(Smooth(DCRL_ThinCond_dcfPhi), Phase)

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 HFSS 3D Layout'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.

Fields Calculator Recipes

The following pages contain calculator recipes for deriving a number of commonly used output parameters from solved HFSS projects.

Calculating Numerical Quantities

Calculating the Current along a Wire or Trace

"Calculating the Voltage Drop along a Line" on page 14-338

"Calculating Net Power Flow through a Surface" on page 14-340

"Calculating the Average of a Field Quantity on a Surface" on page 14-341

"Calculating the Peak Electrical Energy in a Volume" on page 14-342

"Calculating the Q of a Resonant Cavity" on page 14-343

"Calculating the Max Value of a Field Quantity in a Volume" on page 14-346

"Calculating Dielectric (or Conduction) Loss in a Volume" on page 14-347

Calculating Quantities for 2D (Line) Plot Outputs

"Plotting Wave Impedance along a Line" on page 14-347

"Plotting the Phase of E Tangential to a Line/Curve" on page 14-349

"Plotting the Maximum Magnitude of E Tangential to a Line/Curve" on page 14-350

Calculating Quantities for 3D (Surface or Vector) Plot Outputs

"Plotting the E-Field Magnitude Normal to a Surface" on page 14-351

Calculating Quantities for 3D (Volume) Plot Outputs

"Generating an Iso-Surface Contour for a Given Field Value" on page 14-352

Calculating Quantities for Animated Outputs

"Generating an Animation on Multiple Planes with a Positional Variable" on page 14-353

Creating User Defined Named Expressions Library

"Generating Cartesian/Cylindrical/Spherical-Coordinate Field-Components Library" on page 14-354

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Recipe Format

Each calculator recipe will be provided in the format shown below:

EXAMPLE: Title of Current Calculation

Description

The first paragraph will give a brief description of the calculation's intent.

Usage Examples

The second paragraph will give an example of a project type on which the calculation might be useful. It may also comment upon the reasons such a calculation might be of interest.

Prerequisites

The third (optional) paragraph will indicate what must be present before doing the calculator operations, e.g., if certain geometry need to be generated to use in calculations.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Each button click shown as a step	resulting stack entry/type
drop-down menu>Submenu pick also shown as a single step	Scl: {placeholders for numerical results shown in brackets}
Button steps requiring data entry will have entry quantity shown in {brackets}	resulting stack entry/type (*notes follow in italics)*

Calculating the Current along a Wire or Trace

Description

Obtains the full complex current in a wire or trace conductor (e.g., microstrip, stripline) at a specific location by integrating the magnetic field along a closed path encircling the conductor. .

$$I = \oint_{l} \vec{H} \cdot \vec{dl}$$

where *l* is a closed path, which could be a circled 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.

Usage Examples

To find the current distribution along a wire (dipole, monopole, etc.) antenna, this calculation could be repeated at periodic positions along the length of the antenna.

Prerequisites

You must create a closed line for the integration path using **Draw>Line** before beginning calculator operations. The line must be orthogonal to the direction of current flow, should not intersect the wire/trace, and should not be too much bigger than the wire/trace.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>H	CVc : <Hx,Hy,Hz>
Complex>Real	Vec : Real(<Hx,Hy,Hz>)
Geometry>Line...> {select line}	Lin : Line (line1) (user line name may differ from example)
Tangent	SclLin: LineValue(Line(...),Dot(Real<Hx,Hy,Hz>), LineTangent))
∫	Scl : Integrate(Line(....
Complex>CmplxReal	CSc : CmplxR(Integrate(Line(Line1),Dot(…)))
Quantity>H	CVc : <Hx,Hy,Hz>
Complex>Imag	Vec : Imag(<Hx,Hy,Hz>)
Geometry>Line...> {select line}	Lin : Line (line1) (user line name may differ from example)
Tangent	ScLin: LineValue(Line(...),Dot(Imag<Hx,Hy,Hz>), LineTangent))
∫	Scl : Integrate(Line(....
Complex>CmplxImag	CSc : CmplxI(Integrate(Line(Line1,Dot(…)))
+	CSc: (CmplxR(Integrate(Line(Line1),Dot(…))), CmplxI(Integrate (Line(Line1),Dot(…))))
Eval	CSc : {complex numerical value} *(Final complex current result)*

Calculating the Voltage Drop along a Line

Description

Provide the complex voltage drop, in volts between two points by integrating the E-field 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.

$$V = \int_{l} \vec{E} \cdot \vec{dl}$$

where *l* is a path between two points on which voltage difference are measured. Usually it is a straight line object.

Usage Examples

To find the voltage excited across the width of a slot antenna element; to test whether a voltage exceeds breakdown in a particular dielectric media.

Prerequisites

You must create the line along which the E-field is to be integrated using **Draw>Line** before you can complete the calculator routine.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex,Ey,Ez>
Complex>Real	Vec : Real(<Ex,Ey,Ez>)
Geometry>Line...> {select line}	Lin : Line (line1) (*user line name may differ from example*)
Tangent	SclLin: LineValue(Line(...),Dot(Real<Ex,Ey,Ez>), LineTangent))
∫	Scl : Integrate(Line(....
Complex>CmplxReal	CSc : CmplxR(Integrate(Line(Line1),Dot(…)))
Quantity>E	CVc : <Ex,Ey,Ez>
Complex>Imag	Vec : Imag(<Ex,Ey,Ez>)
Geometry>Line...> {select line}	Lin : Line (line1) (*user line name may differ from example*)
Tangent	ScLin: LineValue(Line(...),Dot(Imag<Ex,Ey,Ez>), LineTangent))
∫	Scl : Integrate(Line(....
Complex>CmplxImag	CSc : CmplxI(Integrate(Line(Line1),Dot(…)))
+	CSc: (CmplxR(Integrate(Line(Line1),Dot(…))),CmplxI(Integrate (Line(Line1),Dot(…))))
Eval	CSc : {complex numerical value} (*Final complex voltage result*)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating Net Power Flow through a Surface

Description

This recipe allows calculation of power flow through an open or closed surface by integrating the Poynting vector normal to that surface.

$$W = \int_S Re(\vec{P}) \cdot \vec{n} dS$$

where *S* is the surface that is used to calculate the power, and $\vec{n}$ is the normal vector to the surface *S.*

Usage Examples

This calculation could be used on scattered field data resulting from an incident wave excited HFSS project to evaluate reflection from a radome filter or FSS (frequency selective surface). It might also be used on the closed exterior surface of a solid volume to determine power dissipation within the volume (due to conservation of energy, what goes in a closed surface must come out, unless there is a loss or storage [e.g., standing wave, resonance] mechanism).

Prerequisites

The surface on which the integration is to be performed must exist prior you can complete the calculation. If the surface is the exterior of a solid object, no customer geometry creation is necessary. If the surface is only a subset of an object's faces, or a slice through the entire plane of the model not already defined by a separate 2D entity, then you must create a Faces List and/or Cutplane to represent the integration location.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>Poynting	CVc : Poynting
Complex>Real	Vec : Real(Poynting) (*discards the unneeded imaginary component*)
Geometry>Surface...> {select surface}	Srf : Surface(Facelist1) (*above is example; user surface shown may vary*)
Normal	SclSrf : SurfaceValue(Surface(Facelist1), Dot(Real(Poynting), SurfaceNormal) *(takes the dot product of the vector data with the normal to the surface(s) selected*)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
∫	Scl : Integrate(Surface..)
Eval	Scl : {numerical value} (*final answer in **watts***)

Calculating the Average of a Field Quantity on a Surface

Description

This recipe permits you to calculate the average of a field quantity on a Surface geometry, by dividing the Integration of the field value on the surface by the surface area.

Usage Examples

This calculation could be used to determine the average phase of the E-field at a given cutplane through a project, to find the average current on a trace surface, or to calculate the average H-field tangential to a 2D object used as an aperture. The specific example steps below will be for the first usage example mentioned (average phase of an E-field on a surface), but the format for integration on a surface and for finding the area of the surface is identical for the other applications as well.

Prerequisites

The surface on which the integration is to be performed must exist. If the surface is the exterior of a solid object, no customer geometry creation is necessary. If the surface is only a subset of an object's faces, or a slice through the entire plane of the model not already defined by a separate 2D entity, then you must create a Faces List and/or Cutplane to represent the integration location.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>{select field quantity}	CVc : <Ex, Ey, Ez> (E-field used as example)
{Derive desired scalar field data For example: Scal?>ScalarX Complex>CmplxPhase}	CSc : ScalarX(<Ex, Ey, Ez>) *(first operation result)* Scl : Phase(ScalarX(<Ex, Ey, Ez>)) *(second operation result)*
Geometry>Surface...>{select surface}	Srf: Surface(plane1) (*user surface shown may vary*)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Value	SclSrf : SurfaceValue(Surface(plane1), Phase(ScalarX (<Ex, Ey, Ez>)))
∫	Scl : Integrate(...)
Geometry>Surface...>{select surface}	Srf : Surface(plane1)
Unit Vec>Normal	Vec : SurfaceNormal
Geometry>Surface...>{select surface}	Srf : Surface(plane1)
Normal	SclSrf: SurfaceValue(Surface(plane1)... (*takes the dot product of the surface with its own normal*)
∫	Scl : Integrate(Surface(...
/	Scl : /(Integrate(SurfaceValue(...
Eval	Scl : {numerical value} (*for this example units are in* ***deg*** *or* ***radians***)

Calculating the Peak Electrical Energy in a Volume

Description

This recipe permits you to calculate the peak electrical energy in a volume object. The solution is achieved by integrating **E E** within the volume.

$$W = \int_V \frac{\epsilon_r \epsilon_0 (\vec{E} \cdot \vec{E}^*) dV}{2}$$

where *V* is the volume.

Usage Examples

This calculation could be used to determine the average total energy with respect to time in a terminating resonant cavity. (In a sealed, one-port structure at resonance, energy is converted back and forth between the electrical and magnetic fields, but maintains the same total quantity; therefore, the peak electrical energy is equal to the average total energy.)

Prerequisites

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The volume object which the integration is to be performed must exist before the computation can be completed. If the volume for integration consists of the volume of several drawing objects, you must create a single list entry representing their combined volumes using **Modeler** > **Named Selection** > **Create** > **Object Selection**.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez>
Complex>Conj	CVc : Conj(<Ex, Ey, Ez>)
Quantity>E	CVc : <Ex, Ey, Ez>
Dot	CSc : Dot(Conj(<Ex, Ey, Ez>), <Ex...
Complex>Real	Scl : Real(Dot(Conj(<Ex, Ey, Ez>), ... (*note: the dot product of the E with its conjugate should lead to a real quantity, but the calculator still assume as complex*)
Geometry>Volume...> {select volume}	Vol : Volume(box1) (*above is example, user entry may differ*)
∫	Scl : Integrate(Volume(...
Eval	Scl : {numerical quantity}
Constant>Epsi0	Scl : 8.854187817E-012
Number>Scalar> {enter r for volume}	Scl : {numerical quantity}
*	Scl : {numerical quantity} *(stack entry is volume)*
Number>Scalar>0.5	Scl : 0.5
*	Scl : {numerical quantity}
*	Scl : {numerical quantity} *above is electrical energy in **joules***

Calculating the Q of a Resonant Cavity

Description

This recipe permits you to calculate the Q in a homogeneous dielectric-filled cavity with uniform wall losses, using the equation:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Q_u = \frac{\int_{\Omega} |\mathrm{H}|^2 d\Omega}{\frac{s}{2}\oint_{\Gamma} |\mathrm{n} \times \mathrm{H}|^2 d\Gamma + tg\delta\left(\int_{\Omega} |\mathrm{H}|^2\right) d\Omega}$$

where *s* is skin depth, *tg* is dielectric loss tangent, *n* is the surface normal for the cavity wall faces, and and represent wall surface area and cavity volume, respectively.

Usage Examples

To calculate the Q of an air- or solid-dielectric filled cavity, fed with a below-cutoff port aperture, or obtained via an eigen solution.

Prerequisites

The object (or object selection) representing the cavity total volume must already exist, as must the face selection corresponding to the total wall surface area of the cavity. You can create both via the Modeler menu. The solution should be tuned to the desired resonant frequency for evaluation.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>H	CVc : <Hx, Hy, Hz>
Push	*(above entry duplicated)*
Complex>Conj	CVc : Conj(<Hx, Hy, Hz>)
Dot	CSc : Dot(<Hx, Hy, Hz>, Conj(...
Complex>Real	Scl : Real(Dot(<Hx, Hy, ...
Geometry>Volume>{select cavity volume}	Vol : Volume(cav_total) (*above is example; user entry may differ*)
∫	Scl : Integrate(Volume(cav... (*above represents energy stored in cavity*)
Push	*(above entry duplicated)*
Number>Scalar>{enter loss tan for volume}	Scl : {numerical value} (*loss tangent for dielectric fill in cavity*)

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 equation is ***only*** valid for cavities filled with one dielectric material across the entire volume. For cavities with different dielectric fills (e.g., a dielectric resonator within a larger metal cavity), dielectric loss must be evaluated using integration by parts for each dielectric material volume. The equation also assumes the same conductivity for all walls, and no nonreciprocal (e.g., ferrite) property to either walls or fill.

*	Scl : *(Integrate(Volume(... (*above represents energy lost in dielectric material losses*)
Quantity>H	CVc : <Hx, Hy, Hz>
Geometry>Surface>{select cavity surfaces}	Srf : Surface(cav_tot_faces) (*above is example; user entry may differ*)
Unit Vec>Normal	Vec : NormalSurfaceNormal
Cross :	Cross(<Hx, Hy, Hz>, SurfaceNormal)
Push	*(above entry duplicated)*
Complex>Conj	CVc : Conj(Cross(<Hx, Hy, Hz>, ...
Dot	CSc : Dot(Cross(<Hx, Hy, Hz>, ...
Complex>Real	Scl : Real(Dot(Cross(<Hx, ...
Geometry>Surface>{select cavity surfaces}	Srf : Surface(cav_tot_faces)
∫	Scl : Integrate(SurFace(...
Number>Scalar>2	Scl : 2
Constant>Pi	Scl : 3.14159265358979
Function>Scalar>Freq	Scl : {current freq, in Hz}
*	Scl : {numerical result, pi*f}
Number>Scalar>{enter r for walls}	Scl : {entered value, unitless}
*	Scl : {numerical result, pi*f*mur}
Constant>Mu0	Scl : 1.25663706143592E-006
*	Scl : {numerical, pi*f*mur*mu0}
Number>Scalar>{enter wall conductivity}	Scl : {entered value, s/meter}
*	Scl : {numerical, pi*f*mur*mu0* }
	Scl : {numerical, sqrt of above}
*	Scl : {numerical result, 2*above}
1/x	Scl : {numerical result} (*above is skin depth/2*)
*	Scl : *(Integrate(...

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*	Scl : *(Integrate(Volume(... (*above represents energy lost in dielectric material losses*) (*above is energy lost in walls*)
+	Scl : +(*(Integrate(...
/	Scl : /(+(*(Integrate(...
Eval	Scl : {numerical result} (*above is Q of homogeneous fill and wall conductivity cavity, **unitless***)

Calculating the Max Value of a Field Quantity in a Volume

Description

This recipe permits you to calculate the Max electrical field (value or position) in a volume object.

Usage Examples

This calculation could be used to determine the Max (or Min) value (or position) of electrical field in a resonant cavity or dielectric object. Find out the hot (or quiet) spot value and location.

Prerequisites

The volume (or surface) object within which the Max function is to be performed must already exist before the computation can be completed. If the volume (or surface) for Max function consists of the volume of several drawing objects, you first create a single list entry representing their combined volumes using **Modeler>List>Create>Object (or Face) List**.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Named Expressions>Mag_E>Copy to stack	Scl : Mag_E (Mag_E *is used as an example*)
Geometry>Volume..> {select volume}	Vol : Volume(box1) (*above is example, user entry may differ*)
Max>Value	Scl:Maximum(Volume(box1),Mag_E) *(user can enter Max→Position to find out Max E location. Also Min→Value (Position) leads to* Min E value and location)
Eval	Scl : {numerical quantity}

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating Dielectric (or Conduction) Loss in a Volume

Description

This recipe shows you to calculate the dielectric/conduction loss in a volume object.

Usage Examples

This calculation could be used to determine the loss in a dielectric or metal object. For loss dielectric or finite conductivity metal object, its loss may have significant impact on user"s design performance. Find out the key loss factors is critical for a successful design.

Prerequisites

The volume object within which the loss calculation is to be performed must already exist before the computation can be completed. If the volume for loss calculation consists of the volume of several drawing objects, you must create a single list entry representing their combined volumes using **Modeler** > **Named Selection** > **Create** > **Object Selection**.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Named Expressions>Volume_Loss_ Density> Copy to stack	Scl : Volume_Loss_Density *(Volume Loss Density is used as an example)*
Geometry>Volume...>{select volume}	Vol : Volume(box1) (*above is example, user entry may differ*)
∫	Scl: Integrate(Volume(box1), Volume_ Loss_...)
Eval	Scl : {numerical quantity} (*above is loss from box1, either dielectric or conductional**, Watt***)

Plotting Wave Impedance along a Line

Description

This recipe generates a 2D plot of wave impedance in ohms vs. length for a line geometry. Wave impedance is obtained directly by taking the ratio of the transverse components of the electric field to the ratio of the transverse components of the magnetic field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Z = \frac{E_o^-(x)}{H_{\bar{o}}(x)}$$

Where $E_o^-(x)$ is the transverse component of the electric field, and $H_{\bar{o}}(x)$ is transverse component of the magnetic field.

Usage Examples

This calculation could be used to display wave impedance vs. position along a length of waveguide with a changing cross-section. It could also be used to display the changes in wave impedance in free space at some boundary (e.g., frequency selective surface, radome) when performed on an incident wave problem.

Prerequisites

The line along which the impedance is to be plotted should be defined before performing this calculation. You can generate a line using **Modeler>Draw>Line**.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez>
Smooth	CVc : Smooth(<Ex, Ey, Ez>)
Complex>CmplxMag	Vec : CmplxMag(Smooth(<Ex, Ey, ...
Number>Vector>{enter unit vector in direction of propagation}	Vec : <0, 0, 1> *(Z-directed unit vector used for example)*
Cross	Vec : Cross(CmplxMag(Smooth(<...
Mag	Scl : Mag(Cross(CmplxMag(Smooth...
Quantity>H	CVc : <Hx, Hy, Hz>
Smooth	CVc : Smooth(<Hx, Hy, Hz>)
Complex>CmplxMag	Vec : CmplxMag(Smooth(<Hx, Hy,...
Number>Vector>{enter unit vector in direction of propagation}	Vec : <0, 0, 1> *(Z-directed unit vector used for example)*
Cross	Vec : Cross(CmplxMag(Smooth(<...
Mag	Scl : Mag(Cross(CmplxMag(Smooth...
/	Scl : /(Mag(Cross(CmplxMag(Sm...
Add	Zwave : /(Mag(Cross(CmplxMag(…
Type: Zwave OK	(*User defined Named Expression Zwave 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.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Done	{*exit field calculator window*}
HFSS>Results> **Create Fields Report>** **Rectangular Plot>** Geometry: Line1 Category: Calculator Expressions Quantity: Zwave New Report	{2D graph displayed} *(y axis is wave impedance in **ohms** and x axis is position along line in drawing units)*

Plotting the Phase of E Tangential to a Line/Curve

Description

This recipe generates a 2D plot of the phase of an E-field whose vector component is tangential to a line. The line may also be a curve (faceted polyline).

Usage Examples

This calculation could be used to display the change in phase of the E field tangential to a circular path within a cylindrical dielectric resonator, when used on either a driven or eigensolution problem. Identifying the phase change along this curved path is often necessary to determine the mode index (e.g., Mode 10) which a particular eigensolution or S-parameter resonance represents.

Prerequisites

You must define the line along which the phase is to be plotted before performing this calculation. Use the **Modeler>Draw>Line** command.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>E	CVc : <Ex, Ey, Ez>
Geometry >Line >{select desired line}	Lin : Line(line1) (*above line is example; user's may vary*)
Unit Vec>Tangent	Vec : LineTangent
Complex>CmplxReal	CVc : CmplxR(LineTangent) (*converts unit vector to complex vector*)
Dot	CSc : Dot(<Ex,Ey,Ez>),CmplxR(...
Complex>CmplxPhase	Scl : Phase(Dot(<Ex,Ey,Ez>), ...

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Add Type: Ephase OK	Ephase : Phase(Dot(<Ex,Ey,Ez>),… (*User defined Named Expression Ephase is added*)
Done	{*exit field calculator window*}
HFSS>Results> **Create Fields Report>** **Rectangular Plot** Geometry: Line1 Category: Calculator Expressions Quantity: Ephase New Report	{2D graph displayed} *(y axis is E field phase in **deg** and x axis is position along line in drawing units)*

Plotting the Maximum Magnitude of E Tangential to a Line/Curve

Description:

This recipe generates a 2D plot of the maximum magnitude of an E-field tangential to a line. The line may also be a faceted curve. The maximum magnitude is not necessarily tied to the same input phase value along the length of the line.

Usage Example(s):

This calculation could be used to display the maximum magnitude of an E-field at all points along a line or curve in a transmission line structure, where it is the maximum magnitude and not the magnitude along the line corresponding to a single „snapshot in time" (single port excitation phase) that is of interest. Such data could be used to determine whether the present design might exceed dielectric breakdown voltage in a particular location.

Prerequisites:

You should define the line along which the field data is to be plotted before performing this calculation. Use the **Modeler>Draw>Line** command.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>E	CVc : <Ex, Ey, Ez>
Geometry>Line>{select desired line}	Lin : Line(line1) (*above line is example; user's may vary*)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Unit Vec>Tangent	Vec : LineTangent
Complex>CmplxReal	CVc : CmplxR(LineTangent) (*converts unit vector to complex vector*)
Dot	CSc : Dot(<Ex,Ey,Ez>),CmplxR(...
Complex>CmplxPeak	Scl : CmplxMag(Dot(<Ex,Ey,Ex ... (*above quantity is the **maximum** magnitude of the E-field tang. to the line. To obtain the mag. associated with a particular port phase excitation, enter a number into the stack and use the* Complex>AtPhase *operation instead.*)
Add	Et_max : Phase(Dot(<Ex,Ey,Ez>),…
Type: Et_max	(*User defined Named Expression Et_max is added*)
OK	
Done	{exit field calculator window}
HFSS>Results>Create Fields Report>Rectangular Plot	{2D graph displayed} *(y axis is E field mag in **v/m** and x axis is position along line in drawing units)*
Geometry: Line1 Category: Calculator Expressions Quantity: Et_max New Report	

Plotting the E-Field Magnitude Normal to a Surface

Description

This recipe generates a scalar intensity plot of the E-field magnitude normal to a particular surface (or group of object surfaces, list of object faces), relative to a given input phase excitation. where *S* is the surface geometry, and is the normal vector to the surface *S.*

Usage Examples

This calculation could be used instead of the automatic Plot→Fields→MagE upon surface, when only the magnitude of the E-field with a particular vector orientation is desired. For example, to evaluate the field available for coupling to a probe structure with a particular orientation.

Prerequisites

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You should create the plane to which the desired field component should be normal before you begin the following steps.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez>
Function>Phase	Scl : Phase
Complex>AtPhase	Vec : AtPhase(<Ex,Ey,Ez>, Phase)
Geometry>Surface>{select desired cutplane, faces list, or surface list}	Srf : Surface(faces1) *(faces1 used as example)*
Unit Vec>Normal	Vec : SurfaceNormal
Dot	Scl : Dot(AtPhase (<Ex,Ey,Ez>,Phase),…)
Add Type: E_normal OK	E_normal : Dot(AtPhase (<Ex,Ey,Ez>,0),… (*User defined Named Expression E_normal is added*)
Done	{*exit field calculator window*}
Select the surface (or list) **HFSS** > **Fields** > **Plot Fields** > **Named Expression** Select E_normal	{Scalar Plot on faces displayed} *(E-field normal component value in **v/m**)*

Generating an Iso-Surface Contour for a Given Field Value

Description

This recipe generates a geometry entry called an IsoSurface which represents the surface upon which a selected scalar field quantity has a single value. This surface can be displayed, or used in later operations (to plot other quantities upon, etc.).

Usage Examples

This calculation could be used to locate regions of excessive field magnitudes for voltage breakdown or ohmic heating analysis. It could also be used to generate a desired isosurface to be used as an integration surface for another quantity.

Prerequisites

You should plot of the field quantity of interest to determine the isovalue to use. Isovalues should be entered in MKS units (e.g., V/m, A/m) unless the problem is an eigen solution, in which case all field values are normalized to a peak of 1.0.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>E	CVc : <Ex, Ey, Ez> (*IsoSurfaces for other quantities can also be created; E used as example*.)
Smooth	CVc : Smooth(<Ex,Ey,Ez>) (*as this routine generates a surface geometry object, data smoothing is recommended*)
Function>Phase	Scl : Phase
Complex>AtPhase	Vec : AtPhase(Smooth(<Ex,Ey,Ez>),Phase)
Scal>ScalarX	Scl : ScalarX(AtPhase(Smooth(<Ex,...
Add... Type: E_x OK	E_x: ScalarX(AtPhase(Smooth<Ex,… (E_x *is used here as an example; you can apply other quantity*)
Done	{*exit field calculator window*}
Highlight the geometry	{*Scalar Plot in the geometry displayed*}
HFSS>Fields>Plot Fields>Named Expression Select E_x	*(E-field x-component value in **v/m**)*
HFSS>Fields>Modify Plot Attribute… Select the plot OK **Scale** tab: Num. Division: *1* Use Limits: *100 (as an example)* **Plots** tab IsoValSurface checked Apply	{*IsoSurface contour is displayed*}

Generating an Animation on Multiple Planes with a Positional Variable

Description

This recipe generates animated field output in which each frame is a snapshot of the fields on a different plane of the modeled volume. Any derived field quantity could be plotted in this manner, but this example will simply use the E-field magnitude at zero degrees input excitation.

Usage Examples

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This calculation permits you to generate animated output results in addition to those automatically available from the post-processor. For example, peak E field (E dot E conjugate) could be plotted at multiple planes in sequence.

Prerequisites

This operation will only work in the global coordinate system if you are using X, Y, or Z positions as the animation variable.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez> (*Animations for other quantities can also be created; E used as example*.)
Smooth	CVc : Smooth(<Ex,Ey,Ez>)
Number>Scalar>”0”	Scl : 0
Complex>AtPhase	Vec : AtPhase(Smooth(<Ex,Ey,Ez>,.
Mag	Scl : Mag(AtPhase(Smooth(<Ex,...
Add Type: E_mag0 OK	E_mag0 : Dot(AtPhase(<Ex,Ey,Ez>,0),… (*User defined Named Expression E_mag0 is added*)
Done	{*exit field calculator window*}
Planes>Global:YZ *(user can choose other planes Under modeler tree/Planes)*	(*YZ plane that can vary with X position)*
HFSS>Fields>Plot Fields>Named Expression (E_mag0)	*(Plot named expression on YZ plane)*
HFSS>Fields>Plot Animate>New	{*launches Animation Plot Settings*}
Swept variable Normalized Distance OK	{*displays animation*}

Generating Cartesian/Cylindrical/Spherical-Coordinate Field-Components Library

Description

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This recipe demonstrates the steps to export user-defined named expressions into a library which can be loaded into and reused in other designs or projects.

Usage Examples

This calculation allows you to generate named expressions in addition to those automatically available ones in the field calculator, and save them as a user library, and reload into other designs/projects for use. For example, Cartesian components of E field are used for demonstration.

Prerequisites

Since this recipe is intended to generate generalized user-defined named-expressions, this operation should not be geometry-related.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Named Expressions>Vector_E	Vec : Vector_E (*E used as example*)
Scal?>ScalarX	Scl : ScalarX(Vector_E)
Add...	
Type inEx	
OK	
Named Expressions>Vector_E	Vec : Vector_E
Scal?>ScalarY	Scl : ScalarY(Vector_E)
Add...	
Type in Ey	
OK	
Named Expressions>Vector_E	Vec : Vector_E
Scal?> ScalarZ	Scl : ScalarZ(Vector_E)
Add...	
Type in Ez	
OK	
Save To…	(*Exyz used as an library name example*)
Select (**Ex, Ey, Ez**)	
OK	
Type in **Exyz** (Library Name)	
Save	

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Load From… *(Find the predefined library)* Open Select (**Ex, Ey, Ez**) OK	*(Named Expressions of Ex, Ey and Ez are loaded into Named Expressions)*

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.

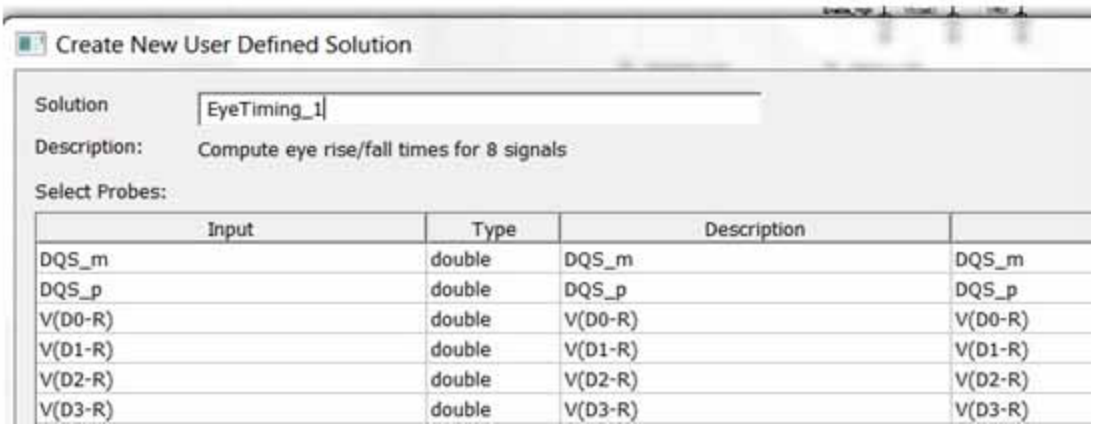
Named Probes and Properties in User Defined Outputs

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.

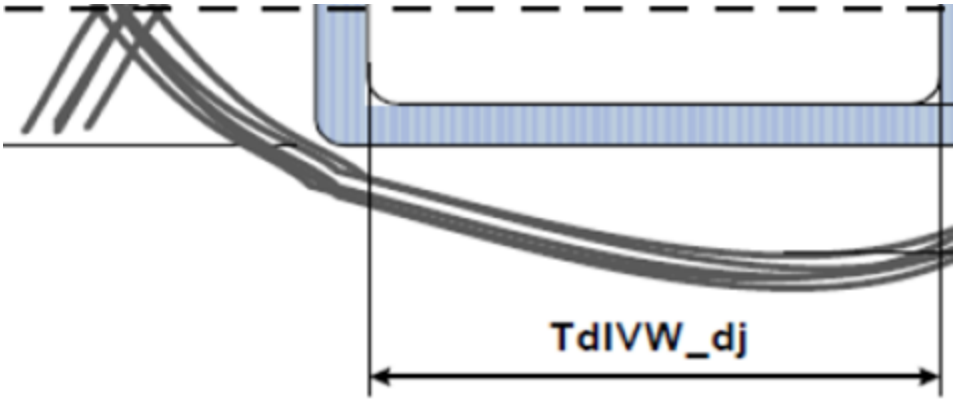
For example, you could create a user-defined solution called EyeTiming_1.

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.

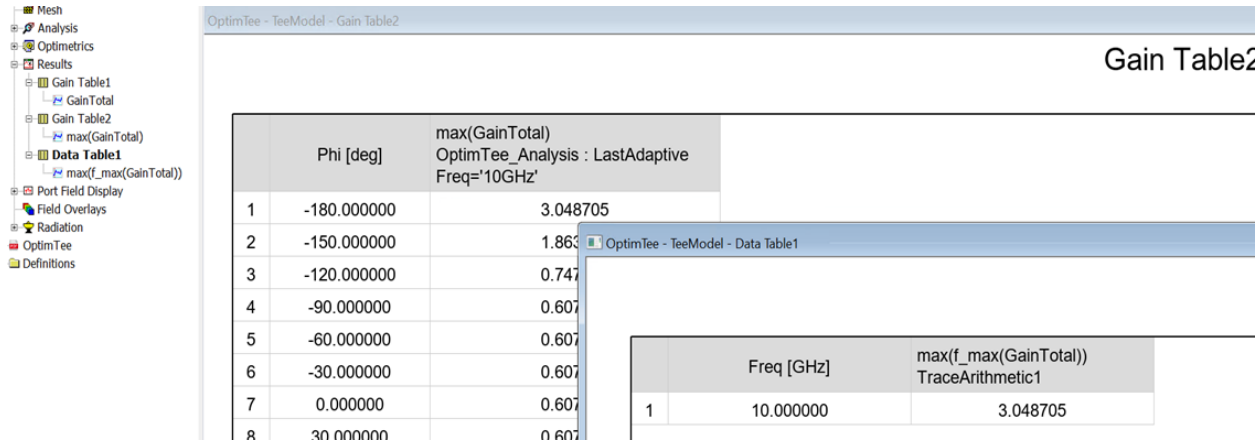


In another example, you could create a user-defined solution called **UDS Distance Trace Arithmetic Result1**:

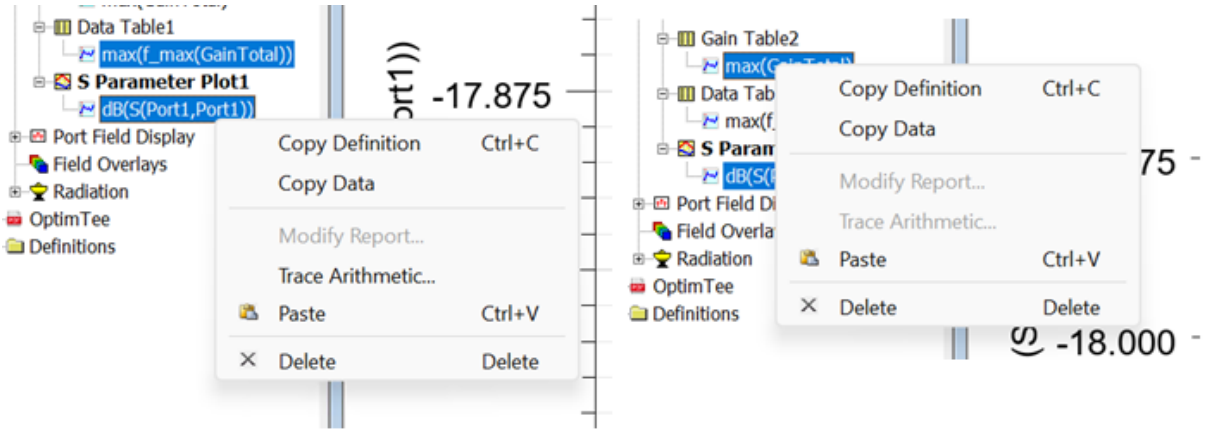
Then access this solution in the **Create User Defined Solution** menu:

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.

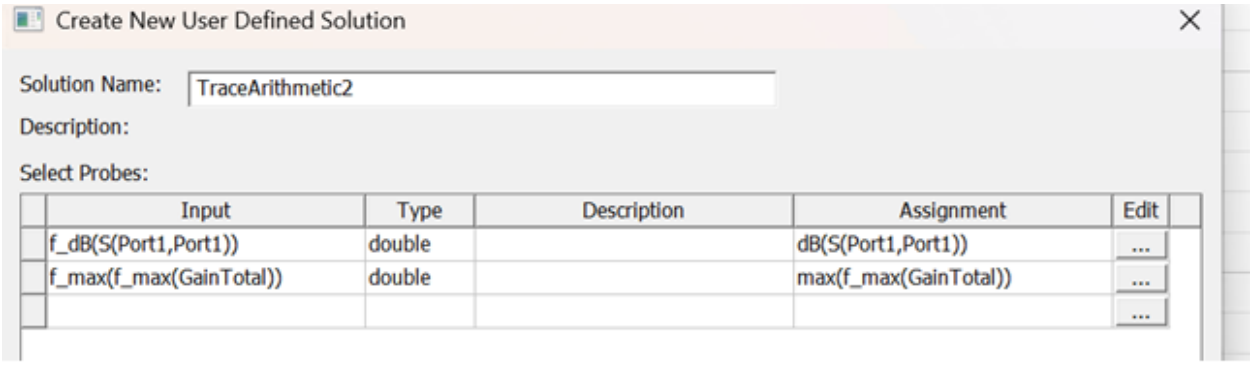


To create a Trace Arithmetic solution, select one (or more) trace and then right click on the selected trace to open the **Trace Menu** window. The Trace Arithmetic menu item is enabled if the selected traces have identical sweep sets.

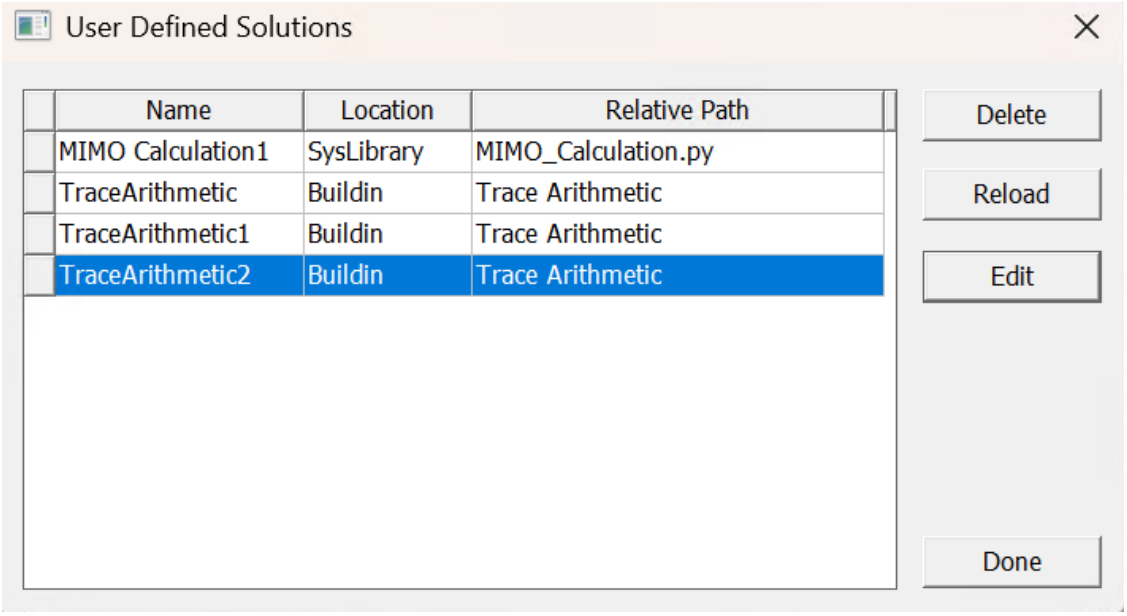


Clicking the Trace Arithmetic item in the **Trace Menu** window brings up the **Create User Defined Solution** window that show the selected traces as the input probes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

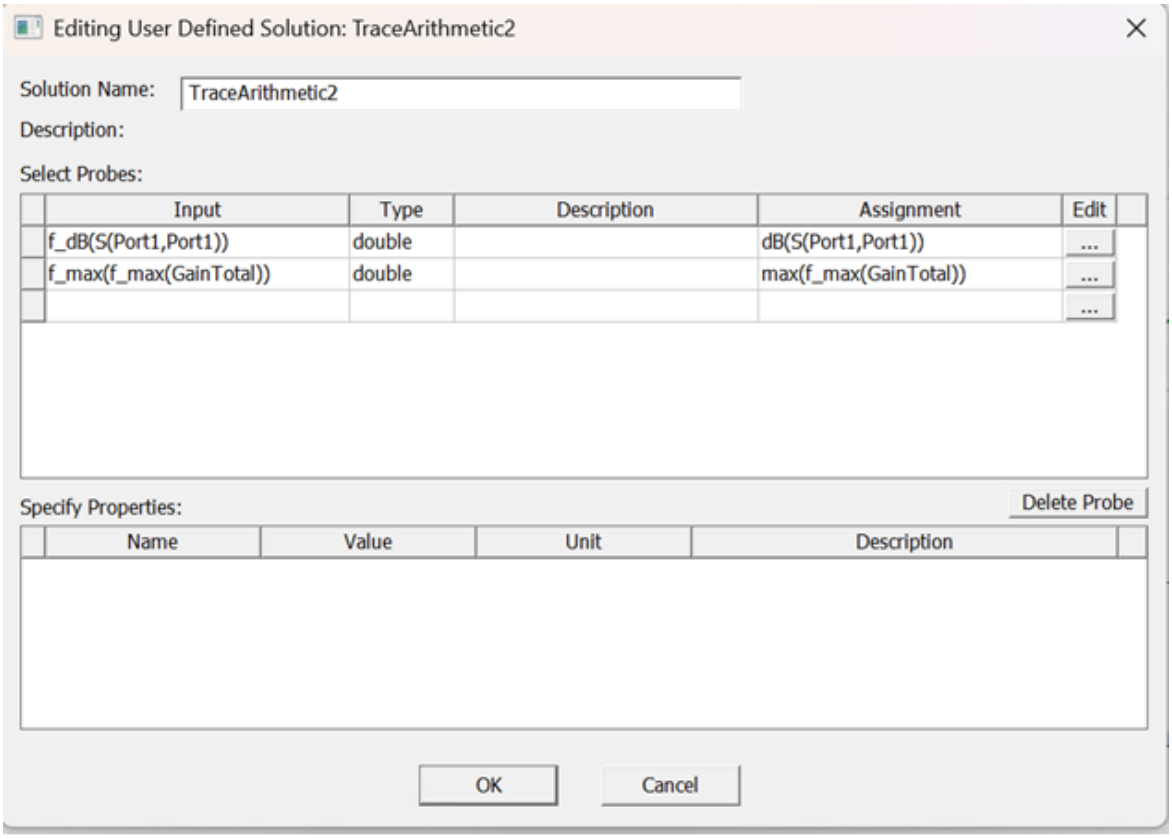


As a Trace Arithmetic solution is a built-in UDS, it can be listed, viewed, and/or edited as a UDS.



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.

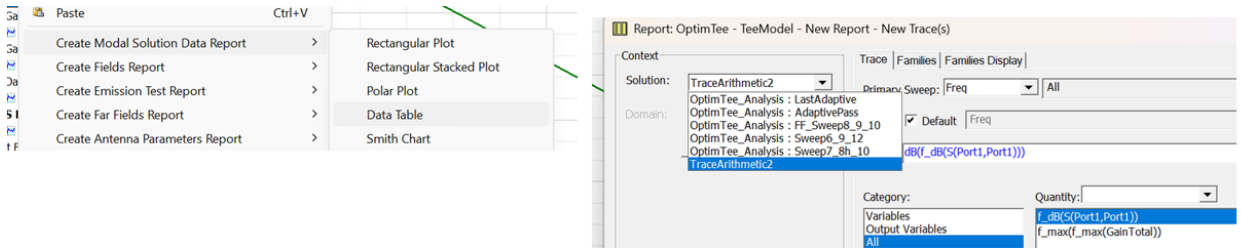
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

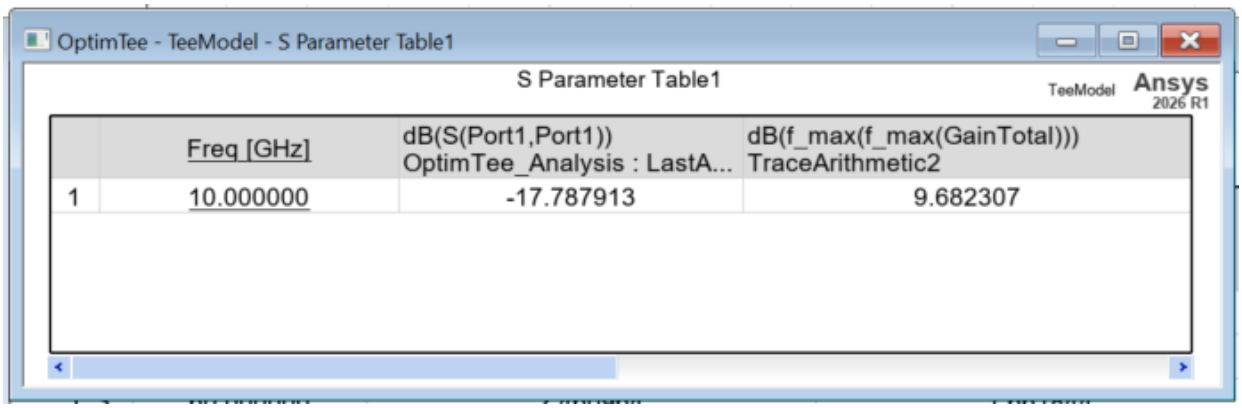


A probe assignment can be an expression using a quantity from import solution, user defined solution, and any report type solutions that have the same sweep set as the first probe.

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 []`

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.

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	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`
	`def GetQuantityNames(self, catName):`
	`  if catName == "UDOOutputs":`
	`  return ["min_val", "max_val", "avg_val"]`
	`  else:`
	`  return []`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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*>)

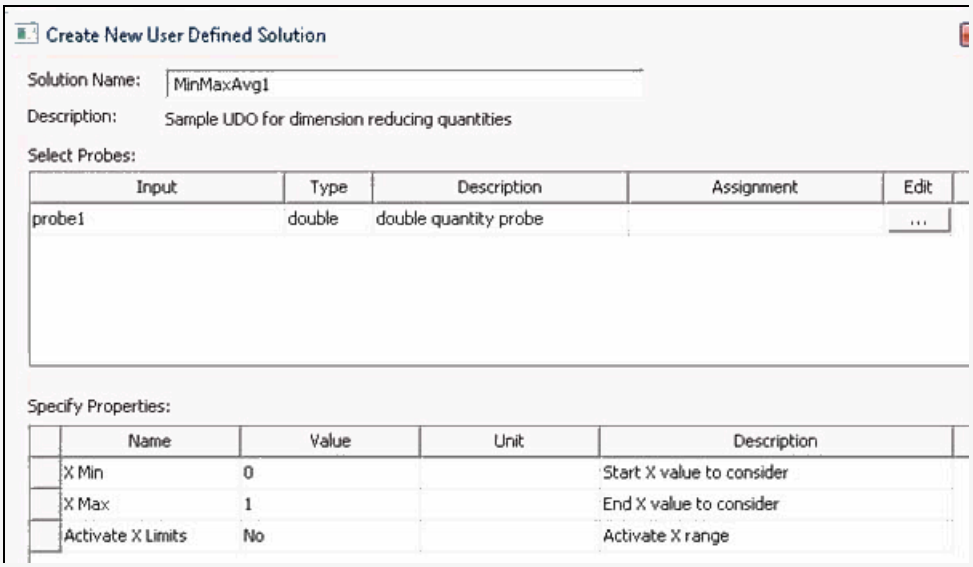
Python Example	`# Returns an instance of QuantityInfo for the qtyName supplied or None if such a`
	`# quantity could not be found`
	`def GetQuantityInfo(self, qtyName):`
	`    # All the quantities we have are simple doubles`
	`    # we can leave them unitless`
	`    return QuantityInfo(Constants.kDoubleParamStr)`

GetInputUDSParams

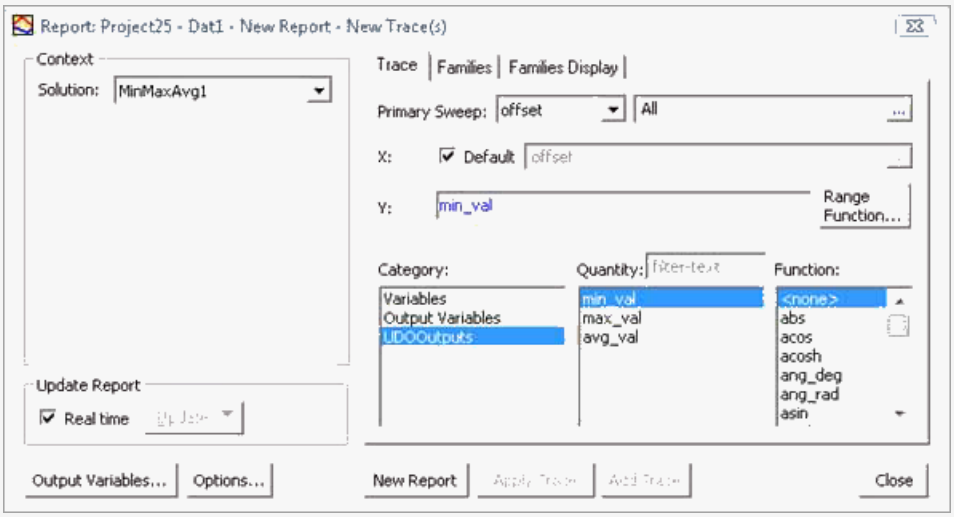
This is the main definition method of the UDO. The supplied arguments populate details of the parameters to which the UDO user will specify value, specify the probe names and their types as well as the dynamic probe selections.

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):`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	pass

Compute

This is the main computation method which generates the data for the quantities that make up the UDO solution.

UI Access	The data is received from the UI using IUDSInputData. It is processed and the result data is sent to the UI using IUDSOutputData.		
Parameters	Name	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:
```


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
```

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **kSweepDomainStr** : string constant to specify the sweep domain
- **kTimeDomainStr** : string constant to specify the time domain

Abstract Classes

- IProgressMonitor
- IUDSInput
- IUDSOutput

IUDSInputData

The purpose of this class is to get data (probe and sweep) from Desktop. The IUDSInputData abstract class declares the following abstract methods:

- GetDoubleProbeData
- GetSweepsDataForProbe
- GetComplexProbeData
- GetSweepNamesForProbe
- GetRequiredQuantities
- 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.		

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	GetDoubleProbeData (<*probeName*>)
Python Example	`# doubleData is a list of floats` `doubleData = inData.GetDoubleProbeData("probe1")`

GetSweepsDataForProbe

All probe data that is supplied is associated with one or more sweep (an intrinsic quantity like Time, Frequency, Theta, Phi, etc. that is swept) quantities.

UI Access	NA		
Parameters	Name <*probeName*> <*sweepName*>	Type String String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO. Name of the sweep.
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.

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	Double array (float in Python) of data for the specified probe. Each pair of floats represents one complex number. The first value is for real part and the second value it the imaginary part. For instance, array [10.0, 0, 5.1, 2.1] represents 2 complex numbers: (10.0, 0) and (5.1, 2.1).

Python Syntax	GetComplexProbeData(<*probeName*>)

Python Example	`# complexDataAsDouble is C# Array of doubles (floats in Python)`
	`# each pair of floats represents one complex number`
	`complexDataAsDouble = inData.GetComplexProbeData ("FarFieldsProbe")`
	`# creating a list of complex numbers from complexDataAsDouble array`
	`complexData = []`
	`if complexDataAsDouble != None:`
	`  for i in xrange(0,complexDataAsDouble.Count, 2):`
	`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*>)

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	`# sweepNames is C# Array of strings` `sweepNames = inData.GetSweepNamesForProbe("FarFieldsProbe")`

GetRequiredQuantities

A given UDO can specify that it provides one or more computed quantities. The user might choose to create a report from only a few among the various available UDO outputs. This function returns the list of UDO output quantities that the user requested. Only these need be computed in the UDO's Compute method.

UI Access	NA
Parameters	None
Return Value	IList<string>

Python Syntax	GetRequiredQuantities()
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`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	`for varPair in theDict:`
	`  varName = varPair.Key #string`
	`  varValue = varPair.Value #string`

GetInterpolationOrdersData

Returns the interpolation orders that are associated with the probe-data. The probe data is specified at each value of the various sweeps. Any value in between the sweep data points, can use the interpolation data to get a possibly more accurate (compared to linear interpolation) inter-sweep value.

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Byte array of interpolation order for the specified probe. These are to be 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`

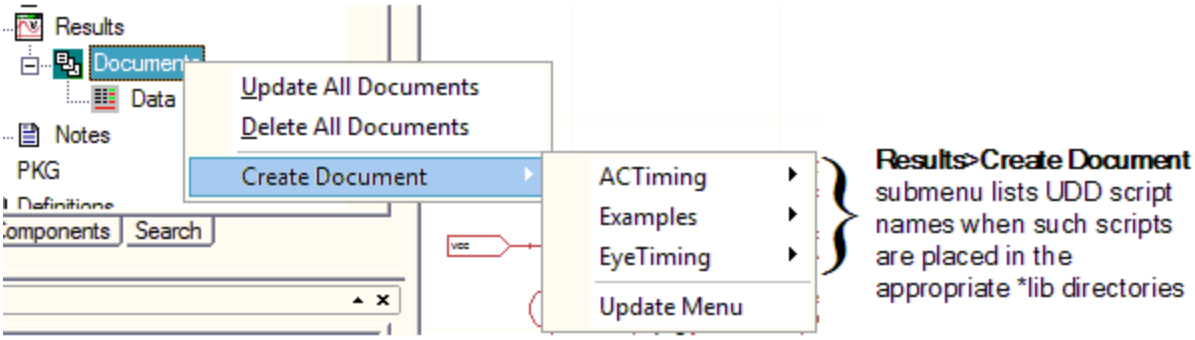
If for some reason you cannot place the .NET assemblies into a Lib directory under *UserDefinedOutputs*, you need to do a couple more steps before step 3 listed above.

```
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.

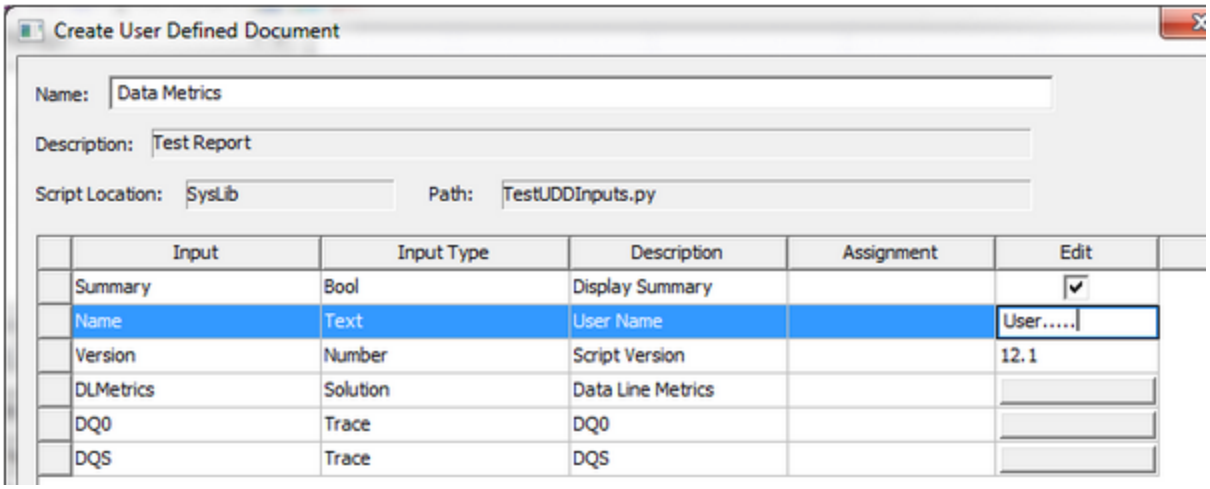
Use **HFSS 3D Layout** > **Results** > **Create Document** > **Update Menu** to refresh the menu to include the new UDD scripts that have been copied to syslib, userlib, or Personallib, or to exclude them if they have been deleted after the launch of desktop.

The UDD scripts that are in syslib/UserDefinedDocuments, userlib/UserDefinedDocuments, or Personallib/UserDefinedDocuments are available through the **HFSS 3D Layout** > **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

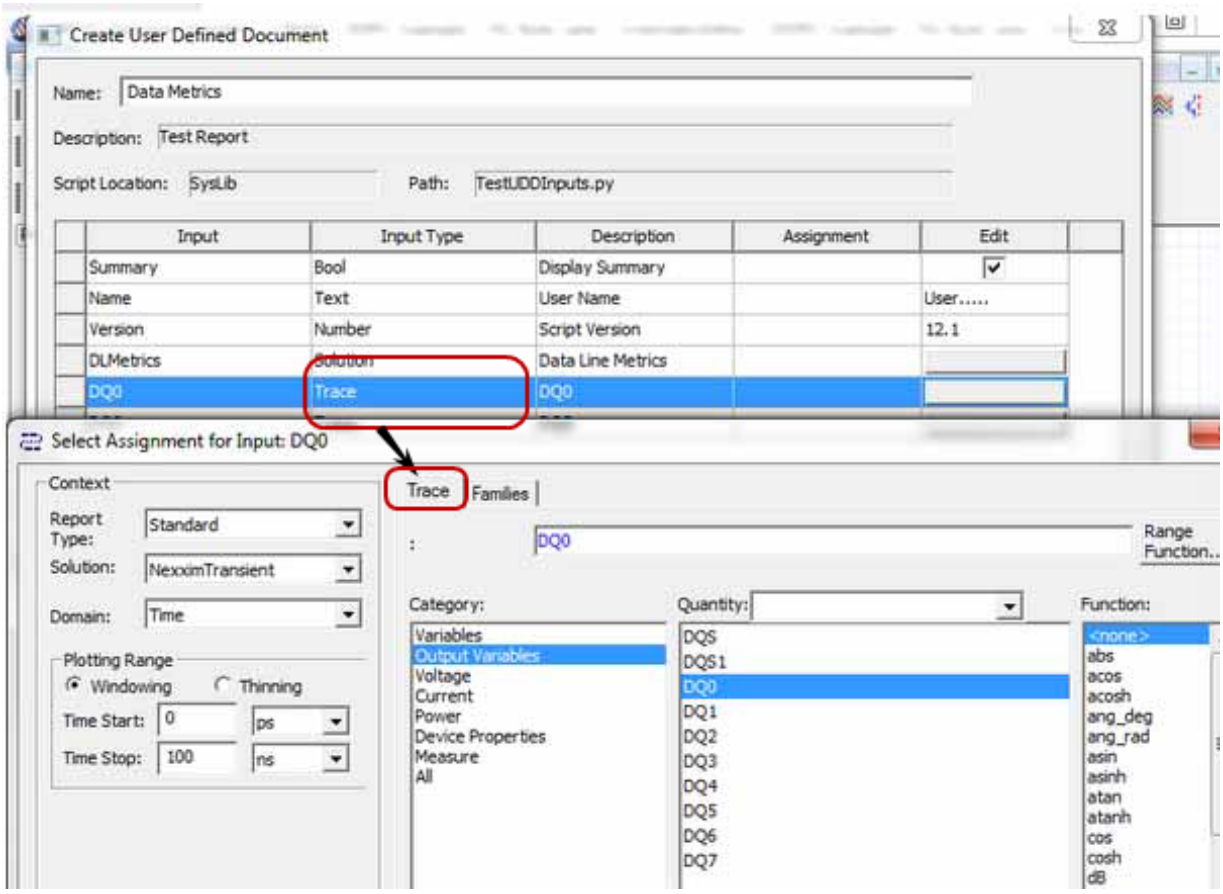
User-defined documents allow data from traces, solutions, and report types as inputs. A UDD can specify the named inputs, which you select or enter the values of in the *Create User Defined Document* dialog box that displays when you run **HFSS 3D Layout** > **Results** > **Create Document** > <*scriptName*>.



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.

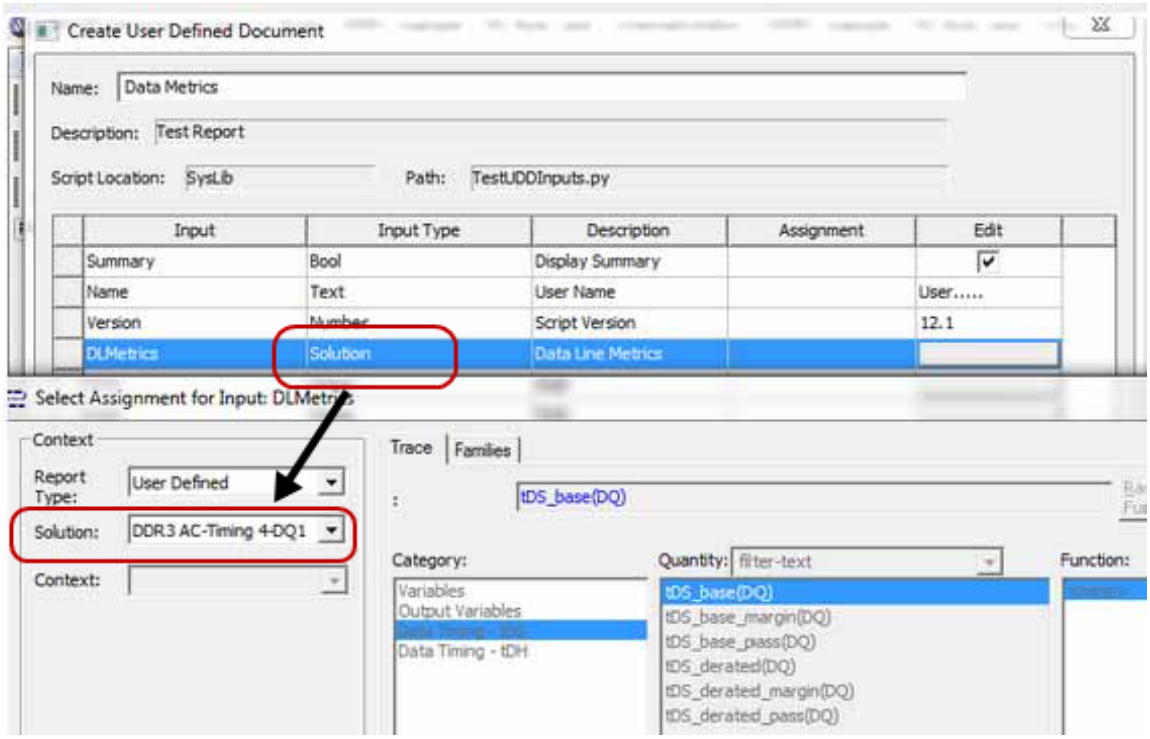


Input Type can also be Solution. You can select an entire solution when the document is created or modified. The solution data in its entirety is now available to the user and can be accessed from the Python script.

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.

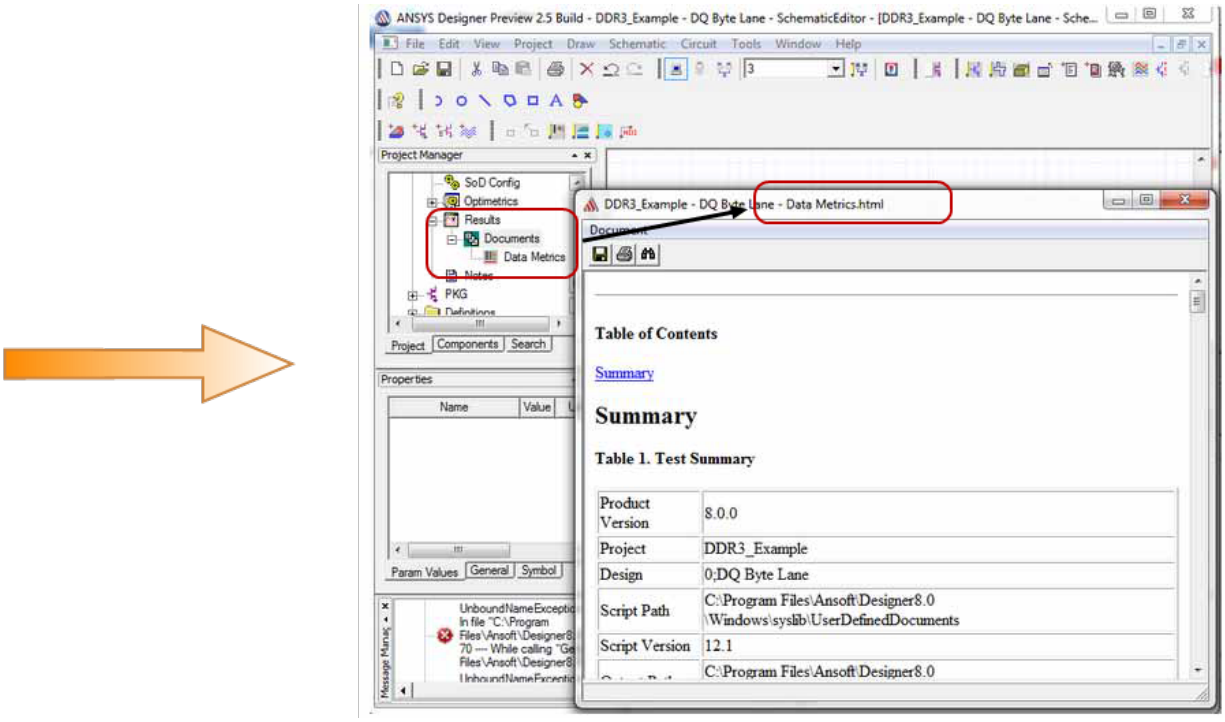


UDD Document Creation and Display

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.

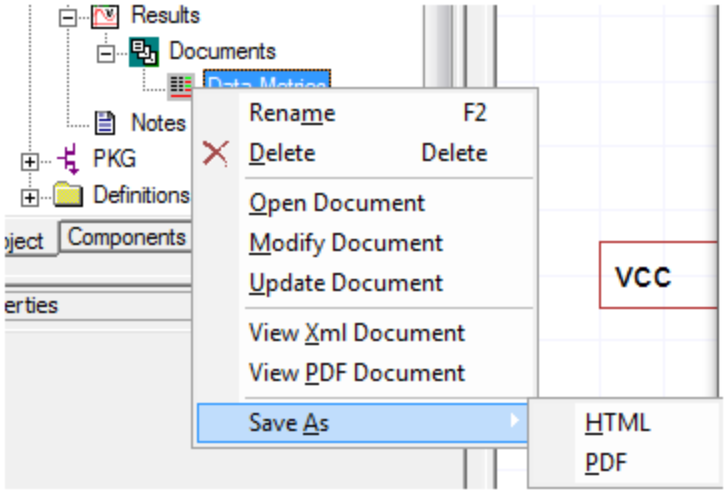
The created document will be placed under a new folder named **Documents** under the **Results** folder. All documents that are created by the user for the design will be placed under this folder.

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

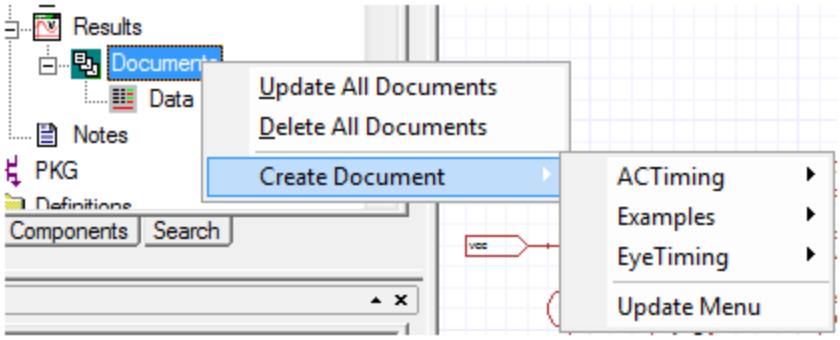
Right-click a user-defined document displayed in the Project Manager tree to bring up a menu where you can rename, delete the document. **Open document** opens the web browser with the html document. **Modify document** opens the setup dialog box, where you can change the selections for the input. To 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.

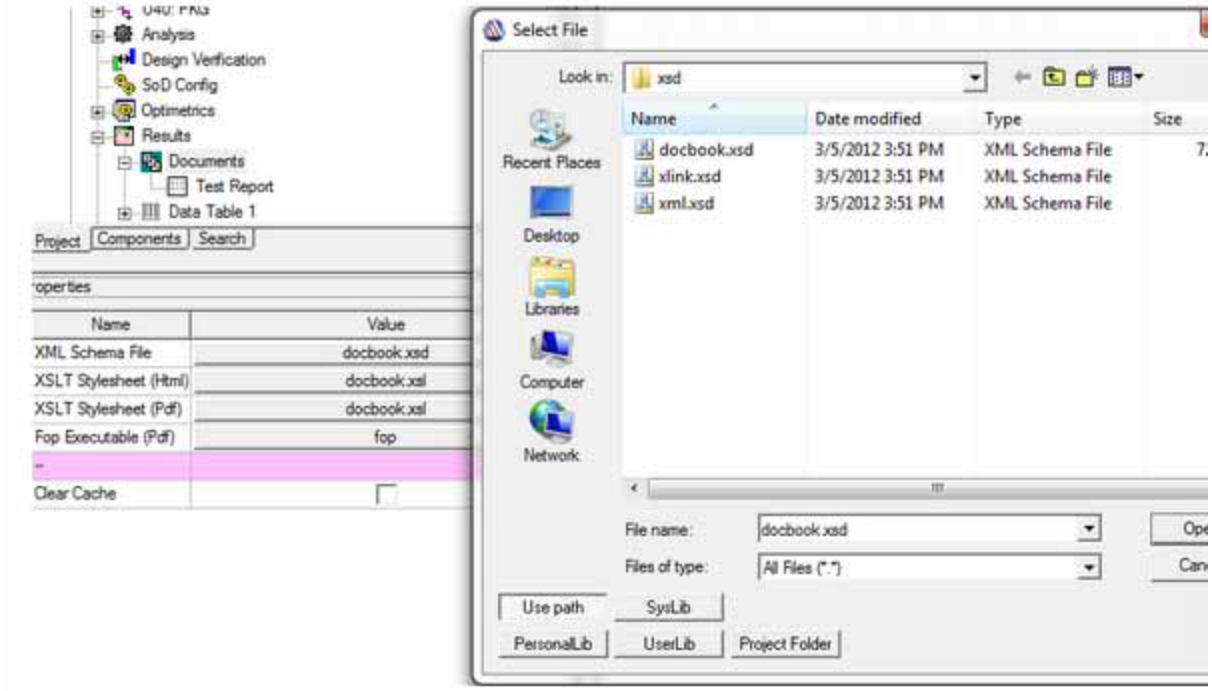


Document Folder's Property Window

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.

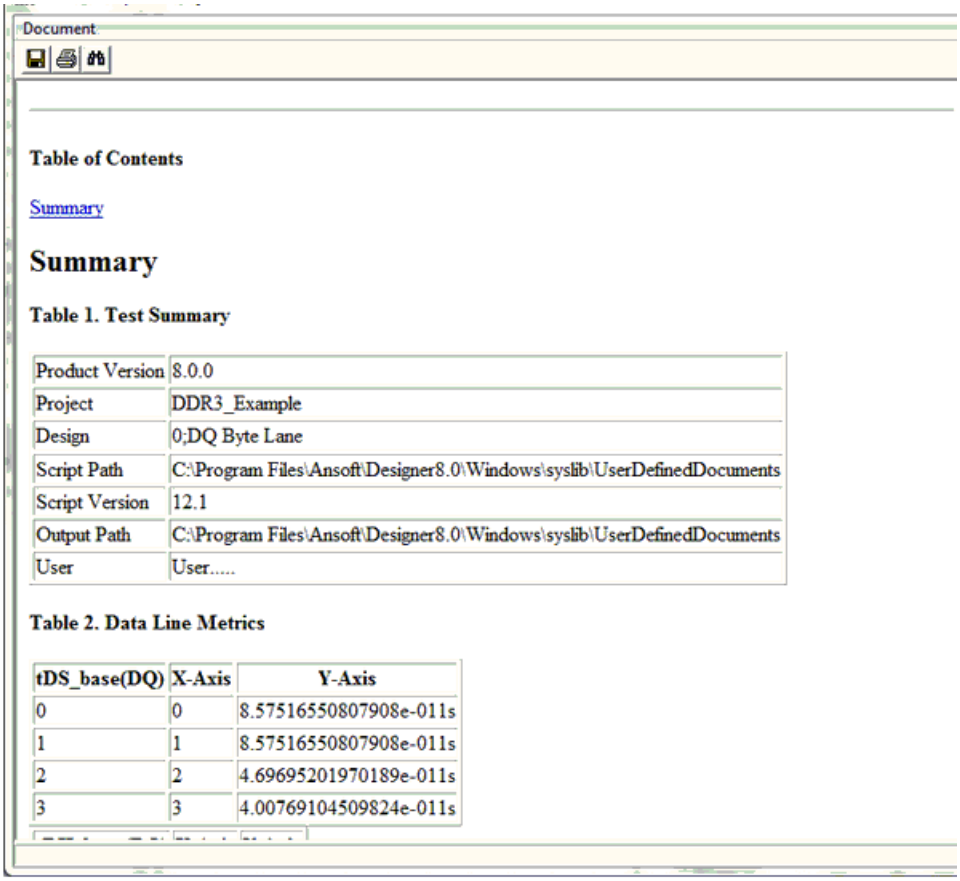

Document

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.

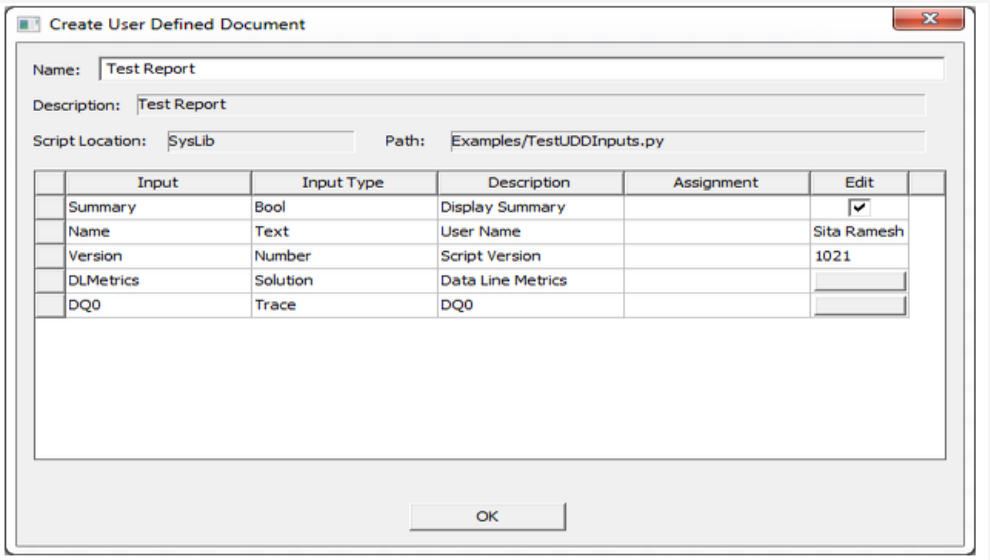
Python Syntax	ShowDefaultSetupDialog()
Python Example	`def ShowDefaultSetupDialog(self):` `  return True`

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 **HFSS 3D Layout**> **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDD Input Interfaces

The Generate function takes in a list of inputs which allows the user to access data from the design.

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.

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): Method used to return x and y double data as a IDictionary<double, double>, given a quantity name.
- 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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scripting objects available in the script for the Generate function:

- oApp = self.GetUDDAppContext()

  Gets the application context. Use this interface to get the active project ad the version of the product.

  ```
  oDesktop = oApp.GetAppDesktop()
    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.

Post-processing and Generating Reports for 2D and Circuit

After HFSS 3D Layout, Nexxim, or Nexsys has generated a solution, all the results are available for analysis. The graphical plots and other reports described in this topic help you to visualize the relationship between design parameters and analysis results.

Creating Reports for 2D and Circuit Projects

A report is generated using the **New Report** window. The appearance of the **New Report** window differs depending on the type of design:

- EM Design (HFSS_3DLayout)
- Circuit Design (Nexxim, Nexsys)

the type of report:

- Standard Report
- Constellation Report
- Eye Diagram Report
- Statistical Eye Report

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 type of display:

- Polar Plot
- Data Table
- Smith Chart
- 3D Polar Plot
- Rectangular Plot
- Smith Contour Plot
- 3D Rectangular Plot
- Rectangular Stacked Plot
- Rectangular Contour Plot

> **Note:** Depending on the nature of your installation and the projects loaded into Electronics Desktop, at times only a subset of the types listed above may appear in the display-types submenu.

When you have selected the display type (e.g., by clicking **Product** > **Results** > **Create Standard Report** > **Rectangular Plot**) a **New Report** window opens that allows you to create the report. The following topics describe how to fill out the **New Report** window for various types of reports and displays.

2D and Circuit Report Types

When you click **Product > Results (** or right-click **Results** in the **Project Tree**), one or more of the following report types and **display types** are available, depending upon the nature of your installation and the projects loaded into Electronics Desktop.

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 Polar Plot • Rectangular Contour Plot

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

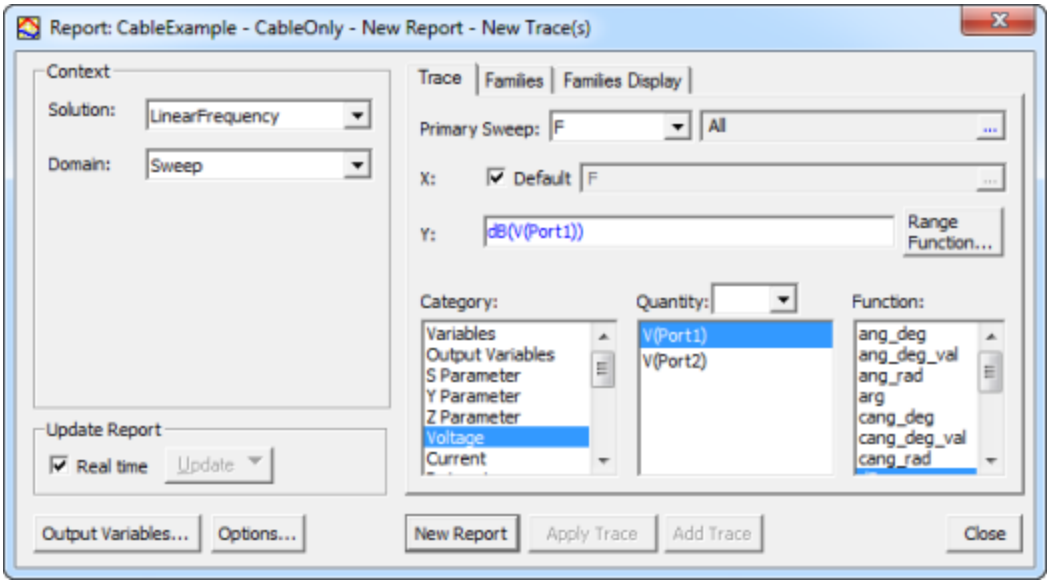
	• Smith Contour Plot
Create Eye Diagram Report	An “Eye Diagram” report can be depicted in any of the following display types: • Rectangular Plot • Data Table • Stacked Eye Diagram Plot
Create Constellation Report	A “Constellation” report can be depicted in any of the following display types: • Rectangular Plot • Data Table
Create Statistical Eye Report	A “Statistical Eye” report can be depicted in any of the following display types: • Rectangular Plot • Data Table
Create Report From File	A “Data File” report can be created in any of the available display types using a previous report that has been saved to a file.

Note: Depending on the nature of your installation and the projects loaded into Electronics Desktop, at times only a subset of the types listed above may appear in the display-types submenu.

2D and Circuit New Report Window

When you click **Product > Results (** or right-click **Results** in the **Project Tree**) and select a **report type** and a **display type** , a **New Report** window similar to the following opens:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: Depending on the nature of your installation and the projects loaded into Electronics Desktop, at times only a subset of the following report options may appear in the **New Report** window.

In the **New Report** window, make selections in the following group boxes to create a report:

Context Group Box

Context group box

- **Solution** – lists the available sweeps.
- **Domain** – lists the available domains.
- **Show** — lists the quantity types to display. (Differential Pairs only)

Harmonics Group Box

- For Harmonic Balance simulation setups (HB1Tone, HBNtone, Oscillator, OscillatorNTone, PXF, TVNoise) this box displays the first five harmonic quantities of a Sweep or Network Function domain.
- **Edit** – You can select/filter and change the displayed harmonic quantities by clicking **Edit** to open the output quantity window; any changes that are made are reflected in the original setup window.

Plotting Range Group Box

- Long-time domain simulations may contain extremely large amounts of data.
- **Windowing** – shows data within a range of times:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Time Start — allows you to set the start time and units.
- Time Stop — allows you to set the stop time and units.
- Note that the maximum number of points displayed is 20,000,000. This may limit the range that can be shown. For example a Transient Analysis with 10ms Stop Time and 1ns Step Time creates 100,000,000 points; only a subset of these (2ms worth) can be displayed. To see the entire range, increase the Step Time.

- **Thinning** — removes unnecessary detail:
  - dy/dx tolerance — a delta voltage that should be set to be smaller than the overall voltage swing (min, max) and greater than the voltage fluctuations to be ignored. This number is used to calculate slope variations and eliminates points that have equal slopes (in these variations). For square or triangular waves, this can limit the number of points to two per group box of constant slope.
  - Number of Points gives the maximum points displayed and is applied after the dy/dx tolerance. It must be less than the maximum number of points.
- **Time Start** – allows you to set the start time and units.
- **Time Stop** — allows you to set the stop time and units.

Constellation Trace Tab

- **Primary Sweep** — drop-down menu allows you to select a primary sweep, or browse for the selection by clicking **[...]**.
- **Constellation** — displays the applicable function. Contains a drop-down menu for selecting the value(s) and a **[...]** browse button for selecting from a list of sweeps (if available). Clicking **Range Function** opens the **Set Range Function** window where a different function can be chosen and applied apply.
- **Category** window- these depend on the Solution type and the design. This lets you specify the category of information for the report.
- **Quantity** – allows you to specify a quantity for the **Constellation** report.
- **Function** – allows you to specify a function for the **Constellation** report.

Eye Diagram Trace Tab

- **Primary Sweep** — drop-down menu allows you to select a primary sweep, or browse for the selection by clicking **[...]**
- **Eye Diagram** — displays the applicable function. Contains a drop-down menu for selecting the value(s) and a **[...]** browse button for selecting from a list of sweeps (if available). Clicking **Range Function** opens the **Set Range Function** window and allows you to choose a different function to apply.
- **Category** window- these depend on the Solution type and the design. This lets you specify the category of information for the report.
- **Quantity** – allows you to specify a quantity for the **EyeDiagram** report.
- **Function** – allows you to specify a function for the **EyeDiagram** report.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constellation Parameters Group Box

- Radio buttons allow you to choose between **Vector** and **Constellation**
- When **Constellation** is selected, you may specify a **Sample Period** and **Offset**.

Eye Parameters Group Box

- Allows you to specify a **Unit Interval** and **Offset**.

Update Report Group Box

- **Real Time** checked — enable real time updates for all reports.
- **Real Time** unchecked — enables the adjacent dropdown menu which allows you to choose **Update This Report** or **Update All Reports**. Both selections update traces to the latest data available.

Trace Tab

- **Primary Sweep** — drop-down menu allows you to select a primary sweep, or browse for the selection by clicking the **[...]**
- **X:**
  - Contains a drop-down menu for selecting the value(s) and a **[...]** button for selecting from a list of sweeps (if available).
- **Y**:
  - **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
  - **Category** window— these depend on the **Solution** type and the design. This lets you specify the category of information for the Y component.
  - **Quantity** – allows you to specify a quantity for the Y component.
  - **Function** – allows you to specify a function for the Y component.

Families Tab

- Lists the number of families available.
- Each member of a family defines one point on a curve.
- Radio buttons allow you to choose between **Sweeps** and **Available variations**. Allows you to edit sweeps; **Edit** lets you select variable **Values**.
- **Nominals** field (unavailable if none exist in the design) allows you to choose one of the following:
  - **Set All Variables to Nominal**
  - **Set All Unswept Variables to Nominal**
  - **Choose Nominals**

Families Display Tab

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Allows you to choose on the following types of families display: **AllFamilies**, **Statistics**, or **Histogram**.

Dialog Box Command Buttons

- **Output Variables** – opens the **Output Variables** window.
- **Options** – opens the Report Setup Options window. 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 contains a field for setting the maximum number of significant digits to display for numerical quantities.
- **New Report** – adds a report to the **Project** tree under the **Results** icon. The new report is displayed in the **Project** window.
- **Apply Trace** — applies the currently configured trace.
- **Add Trace** – this is enabled when you have created or selected a report. Add further traces. The new trace displays in the **Project** window under the report.
- **Close** – closes the **New Report** window.

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, S_{11} say, 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(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 so by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val(S_{11}) is 90 and "1 + ang_deg_val(S_{11})" is 91.

2D and Circuit Display Types

The information in a report can be displayed in several formats. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*]. Then right-click **Results**. Navigate to the appropriate report type, then choose from one of the following display types:

Rectangular	A 2D rectangular (x-y) graph.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot	
Polar Plot	A 2D circular chart divided by spherical coordinates.
Data Table	A grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or another quantity.
Smith Chart	A 2D polar chart of S-parameters upon which a normalized impedance grid has been superimposed.
Rectangular Stacked Plot	A 2D rectangular (x-y) stacked graph.
3D Rectangular Plot	A 3D rectangular (x-y-z) graph.
3D Polar Plot	A 3D circular plot divided by spherical coordinates.
Rectangular Contour Plot	A 2D rectangular contour chart of the eye simulation results.
Smith Contour Plot	A 2D Smith contour chart of the load-pull results.
Eye Diagram Plot	A 2D rectangular-eye diagram with transient data available for post-processing
Stacked Eye Diagram Plot	One or more eye diagram plots in a horizontal stack.
PAM-4 Eye Plot	An eye diagram created from PAM-4 voltage traces with 4 symbol levels that create three distinct sub-eye openings between the three sequential pairs of symbol levels.
Statistical Eye Plot	A 2D rectangular-eye diagram with no retained data for post-processing

> **Note:** Depending on the nature of your installation and the projects loaded into Electronics Desktop, at times only a subset of the types listed above may appear in the display-types submenu.

2D and Circuit Rectangular Plot

A rectangular plot is a 2D, x-y graph of results.

1. Click **Product** > **Results** (or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*]. Then right-click **Results**) and point to a appropriate **report type** . Then select **Rectangular Plot** to open the **New Report** 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. In the **Context** group box make selections on 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** and **display type** selected.
    c. Show field with a drop down selection list. Whether this field appears depends on if differential pairs exist and are enabled.
3. Under the **Trace** tab **Primary Sweep** group box, make selections on the following categories:
    a. In the **Category** group box, click the type of information to plot.
    b. In the **Quantity** group box, 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. Edit this field directly. Note that invalid expressions are displayed in red.
    e. **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
4. In the **X (Primary Sweep)** group box, make selections for the following:
    a. Select the Primary value(s) on the dropdown menu.
    To select an X component that is different on the Primary Sweep, uncheck the Default field to enable the X field and **[...]** button. Click **[...]** to open the **Select X Component** window. 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 window to assign the X component.
    b. If sweeps are available, select **[...]** to open a window that lets you select particular sweep or sweeps, or all sweeps.
    c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (e.g., 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 **[...]** button.
5. From the **Families** tab, confirm or modify the sweep variables that is 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** window.

    The function of the selected quantity is plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the **Project Manager** window and the traces are listed under the plot. When you select the traces or

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

2D and Circuit Polar Plot

In Ansys Electronics Desktop, a polar plot is a 2D circular chart divided by the spherical coordinates R and theta, where R is the radius, or distance on the origin, and theta is the angle on the x-axis. Following is the general procedure for drawing a polar graph of results:

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** . Then select **Polar Plot** top open the **New Report** window.
2. In the **Context** group box make selections on 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** and **display type** selected.
    c. Show field with a drop down selection list. Whether this field appears depends on if differential pairs exist and are enabled.
3. In the **Trace** tab **Polar Component** group box, specify the information to plot:
    a. From the **Category** dropdown menu, select the type of information to plot.
    b. From 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. Edit this field directly. Note that invalid expressions are displayed in red.
    e. **Range Function** — opens the **Set Range Function** window. 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** window.

The function of the selected quantity is plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the **Project Manager** window and the traces are listed under the plot. When you select the traces or plots, their

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

2D and Circuit Data Table

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 **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*]. Then right-click **Results**. Navigate to the appropriate **report type** . Then select **Data Table** to open the **New Report** window.
2. In the **Context** group box make selections on the following field or fields, depending on the design and solution type.
    a. Solution field with a drop down selection list.
    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** and **display type** selected.
    c. Show field with a drop down selection list. Whether this field appears depends on if differential pairs exist and are enabled.
3. Under the **Trace** tab **Primary Sweep** group box, make selections on the following categories:
    a. From the **Category** drop-down menu, select the type of information to plot.
    b. From the **Quantity** group box, click the values to plot. Use **Ctrl**+click to make multiple selections.
    c. In the **Function** group box, click the mathematical function to apply to the quantity for the plot.
    d. The **Value** field displays the currently specified Quantity and Function. Edit this field directly. Note that invalid expressions are displayed in red.
    e. **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
4. In the **X (Primary Sweep)** group box, make selections for the following:
    a. Select the Primary value(s) on the dropdown menu.
    To select an X component that is different on the Primary Sweep, uncheck the Default field to enable the X field and **[...]** button. Click **[...]** to open the **Select X Component** window. 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 window to assign the X component.
    b. If sweeps are available, select **[...]** to open a window that lets you select particular sweep or sweeps, or all sweeps.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (e.g., 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 **[...]** button.

5. 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** window.

   The Y quantity is listed at each variable value or additional quantity value you specified. The data table is listed under **Results** in the **Project Manager** window. The plot is listed under **Results** in the **Project Manager** window 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.

6. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

2D and Circuit Smith Chart

A Smith chart is a 2D polar plot of S-parameters upon which a normalized impedance grid has been superimposed. Following is the general procedure for creating a Smith chart of results:

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** and select **Smith Chart** to open the **New Report** window.
2. In the **Trace** tab **Polar Component** group box, specify the information to plot:
   a. From the **Category** dropdown menu, select the type of information to plot.
   b. From the **Quantity** list, click the values to plot. Use **Ctrl**+click to make multiple selections.
   c. In the **Function** group box, click the mathematical function to apply to the quantity for the plot.
   d. The **Value** field displays the currently specified Quantity and Function. Edit this field directly. Note that invalid expressions are displayed in red.
   e. **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
3. 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** window.

The function of the selected quantity is plotted against the values you specified on a polar plot. Each circle on the plot is labeled with values of *R*, measuring normalized resistance, and each line is labeled with values of *X*, measuring normalized reactance. The plot is listed

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

under **Results** in the **Project Manager** window 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.

4. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**.

2D and Circuit Rectangular Stacked Plot

A rectangular stacked plot is a 2D stacked x-y graph of results.

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** . Then select **Rectangular Stacked Plot** to open the **New Report** window.
2. In the **Context** group box make selections on the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list.
   b. Show field with a drop down selection list. Whether this field appears depends on if differential pairs exist and are enabled.
3. Under the **Trace** tab **Z Component** group box, specify the information to plot along the z-axis:
   a. In the **Category** list, click the type of information to plot.
   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. Edit this field directly. Note that invalid expressions are displayed in red.
   e. **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
4. From 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 on the drop down list.
- If sweeps are available, select the **[...]** button to display a window that lets you select particular values. The quantity is plotted against the primary sweep variable listed.

5. In the **X (Primary Sweep)** group box, make selections for the following:
   a. Select the Primary value(s) on the dropdown menu.
   To select an X component that is different on the Primary Sweep, uncheck the Default field to enable the X field and **[...]** button. Click **[...]** to open the **Select X Component** window. 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 window to assign the X component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If sweeps are available, select **[...]** to display a window that lets you select particular sweep or sweeps, or all sweeps.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (e.g., 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 **[...]** button.

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** window.

The function of the selected quantity or quantities is plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the **Project Manager** window. 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**.

2D and Circuit 3D Rectangular Plot

A rectangular plot is a 3D, x-y-z graph of results.

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** . Then select **3D Rectangular Plot** to open the **New Report** window.
2. In the **Context** group box make selections on the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list.
   b. Show field with a drop down selection list. Whether this field appears depends on if differential pairs exist and are enabled.
3. Under the **Trace** tab **Z** Component group box, specify the information to plot along the z-axis:
   a. In the **Category** group box, click the type of information to plot.
   b. In the **Quantity** group box, 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. Edit this field directly. Note that invalid expressions are displayed in red.
   e. **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
4. From 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.

- From the drop-down menu, select the sweep variable to use.
- If sweeps are available, select the **[...]** button to display a window that lets you select particular values. The quantity is plotted against the primary sweep variable listed.

5. In the **X (Primary Sweep)** group box, make selections for the following:
   a. Select the Primary value(s) on the drop-down menu.
   To select an X component that is different on the Primary Sweep, uncheck the Default field to enable the X field and **[...]**. Click **[...]** to open the **Select X Component** window. This lets you specify the X component as you do the Y; that is, in terms of Categories that define the selectable Quantities, and Functions to apply. After making selections, **OK** the window to assign the X component.
   b. If sweeps are available, select **[...]** to display a window that lets you select particular sweep or sweeps, or all sweeps.
   c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (e.g., 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 **[...]** button.
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** window.

   The function of the selected quantity or quantities is plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the **Project Manager** window. 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**.

2D and Circuit 3D Polar Plot

A 3D polar plot is a 3D circular chart divided by the spherical coordinates R, theta, and phi, where R is the radius, or distance on the origin, theta is the angle on the x-axis, and phi is the angle on the origin in the z direction. Following is the general procedure for drawing a 3D polar plot of results:

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** . Then select **3D Polar Plot** to open the **Report** window.
2. In the **Context** group box make selections on the following field or fields, depending on the design and 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.

a. Solution field with a drop-down menu.
b. Show field with a drop-down menu. Whether this field appears depends on if differential pairs exist and are enabled.

3. In the **Trace** tab **Mag** group box, specify the information to plot along the R-axis, or the axis measuring magnitude:
   a. From the **Category** drop-down menu, select the type of information to plot.
   b. From the **Quantity** group box, 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. Edit this field directly. Note that invalid expressions are displayed in red.
   e. **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
4. From the **Trace** tab **Theta (Secondary Sweep)** line, make a selection on the sweep variable drop-down menu and specify all values or select values to plot along the theta-axis:
5. From the **Trace** tab **Phi (Primary Sweep)** line, select the sweep variable from the drop-down menu, and specify all values or select values to plot along the phi-axis:
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** window.

   The function of the selected quantity or quantities is plotted against the R-, phi-, and theta-axes on a 3D polar graph. The plot is listed under **Results** in the **Project Manager** window. 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**.

2D and Circuit Rectangular Contour Plot

This type of plot is a 2D rectangular contour chart of the eye simulation results. Its traces can have two characteristics for bit error rates: MinEyeHeight and MinEyeWidth.

To display a rectangular contour plot after a successful analysis:

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** . Then select **Rectangular Contour Plot** to open the **New Report** window.
2. Select the output quantities of interest and click **Add Trace**.
3. Click **Done**. A Report window opens to display a graph of the analysis results.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D and Circuit Smith Contour Plot

To display a contour chart of the load-pull results after a successful analysis:

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** and select **Smith Contour Plot** to open the **New Report** window.
2. Select the output quantities of interest and click **Add Trace**.
3. Click **Done**. A Report window opens to display a graph of the analysis results.

Circuit Eye Diagram Plot

An eye diagram illustrates an overlay of rising and falling bit waveforms with a given unit interval or bit time.

1. Click **Circuit > Results (** or right-click **Results** in the **Project Tree**) , select **Create Eye Diagram Report** > **Rectangular Plot** to open the **New Report** window.
2. In the **Context** group box, make selections on the following 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** and **display type** selected.
    c. **Show** field with a drop down selection list. Whether this field appears depends on if differential pairs exist and are enabled.
3. In the **Eye Parameters** group box, set the **Unit Interval** and **Offset** to match the specification for the source or driver of the signal.
4. Under the **Trace** tab **Primary Sweep** group box, make selections on the following categories:
    a. In the **Category** list, click the type of information to plot. For an Eye Diagram, **Voltage** is the usual Category.
    b. In the **Quantity** list, click the value to plot. The unit interval used to generate the quantity must be the same as the one specified for the report.
    c. In the **Function** list, click the mathematical function of the quantity to plot, typically <none>.
    d. The **Eye Diagram** field displays the currently specified Quantity and Function. Edit this field directly. Note that invalid expressions are displayed in red.
    e. **Range Function** — opens the **Set Range Function** window. This applies to the currently specified Quantity and Function.
5. In the **X (Primary Sweep)** group box, make selections for 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.

a. Select the Primary value(s) on the dropdown menu.
To select an X component other than the Primary Sweep, click the **[...]** button to open the **Select X Component** window. 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 window to assign the X component.

b. If sweeps are available, select **[...]** to display a window that lets you select particular sweep or sweeps, or all sweeps.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (e.g., 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 **[...]** button.

6. From the **Families** tab, confirm or modify the sweep variables that is 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** window.

The function of the selected quantity is plotted against the swept variable values or quantities you specified on an eye diagram. The plot is listed under **Results** in the **Project Manager** window 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.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**. The added trace is overlaid on the existing eye traces.

2D and Circuit Stacked Eye Diagram Plot

A stacked eye diagram creates a multiple display of two or more eye diagrams with the same unit interval bit time. A stacked eye diagram is a convenient way to compare a set of eye diagrams with the same unit interval.

1. Click **Circuit > Results (** or right-click **Results** in the **Project Tree**) , select **Create Eye Diagram Report**, then select **Stacked Eye Diagram Plot** to open the **New Report** window.
2. In the **Context** group box, make selections on 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** and **display type** selected.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

   c. **Show** field with a drop down selection list. Whether this field appears depends on if differential pairs exist and are enabled.

3. In the **Eye Parameters** group box, set the **Unit Interval** and **Offset** to match the specification for the source or driver of the signal. **The unit interval is required.**
4. Under the **Trace** tab **Primary Sweep** group box, make selections on the following categories:
   a. In the **Category** list, click the type of information to plot. For an Eye Diagram, **Voltage** is the usual Category.
   b. In the **Quantity** list, click the value to plot. The unit interval used to generate the quantity must be the same as the one specified for the report.
   c. In the **Function** list, click the mathematical function of the quantity to plot, typically <none>.
   d. The **Eye Diagram** field displays the currently specified Quantity and Function. Edit this field directly. Note that invalid expressions are displayed in red.
   e. **Range Function** — opens the **Set Range Function** window. This applies to the currently specified Quantity and Function.
5. In the **X (Primary Sweep)** group box, make selections for the following:
   a. Select the Primary value(s) on the dropdown menu.
   To select an X component that is different on the Primary Sweep, click the **[...]** button to open the **Select X Component** window. 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 window to assign the X component.
   b. If sweeps are available, select **[...]** to display a window that lets you select particular sweep or sweeps, or all sweeps.
   c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (e.g., 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 **[...]** button.
6. From the **Families** tab, confirm or modify the sweep variables that is 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** window.

The function of the selected quantity is plotted against the swept variable values or quantities you specified on an eye diagram. The plot is listed under **Results** in the **Project Manager** window 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**. The added trace becomes the lowest of the stacked eye diagrams.

> **Note:** The stacked eye diagram is very useful for comparing eye diagrams (e.g., for AMI analysis, stack the waveforms before and after the channel).
>
> - Sweep a variable and stack the resulting set of eye diagrams.
> - When Eye Measurements can be displayed, the Stacked Eye Diagram format displays the measurements in a convenient side column that does not block the chart.

Circuit Statistical Eye Plot

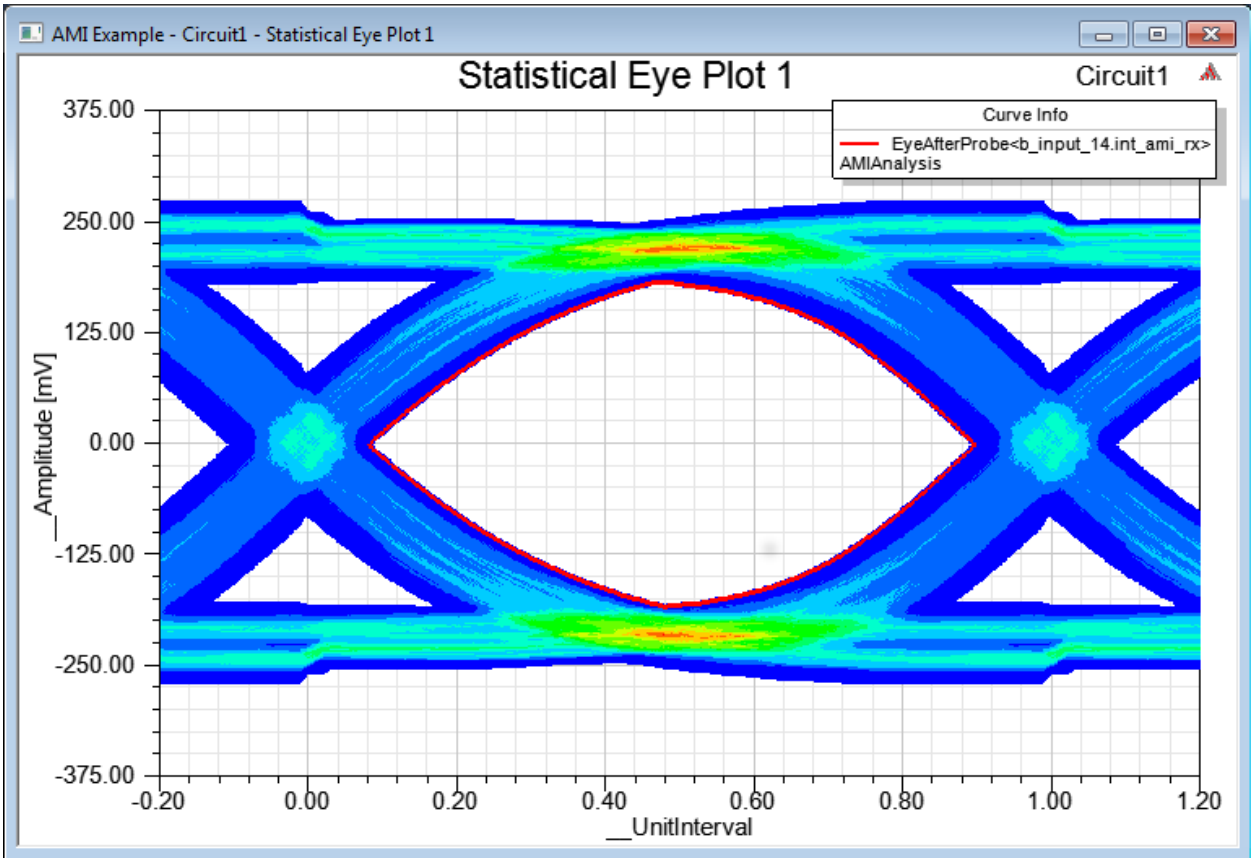
> **Note:** Statistical Eye Plots are intensity plots of Probability Mass Function (PMF) data. For bit-by-bit analyses (transient, QuickEye, AMI) the PMF data is computed on the waveform generated for the given bit pattern, and the inner eye contour of the plot corresponds to the worst-case eye for that bit pattern. *Inner eye contours on Statistical Eye Plots are not BER contours.* Contour plots provide BER contours.

To display a statistical eye plot of the transient, QuickEye, or AMI results after a successful analysis:

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type**. Then select **Statistical Eye Plot** to open the **New Report** window.
2. Select the output quantities of interest and click **Add Trace**.
3. Click **Done**. A Report window opens to display a graph of the analysis results.

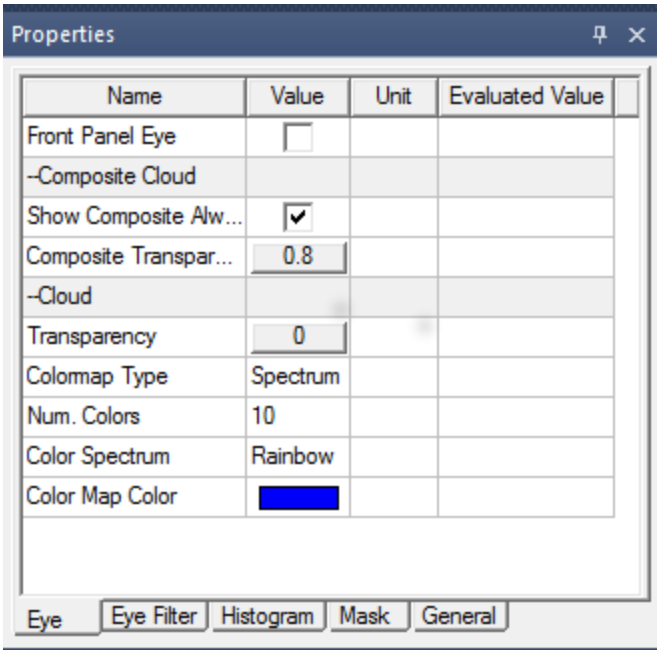
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

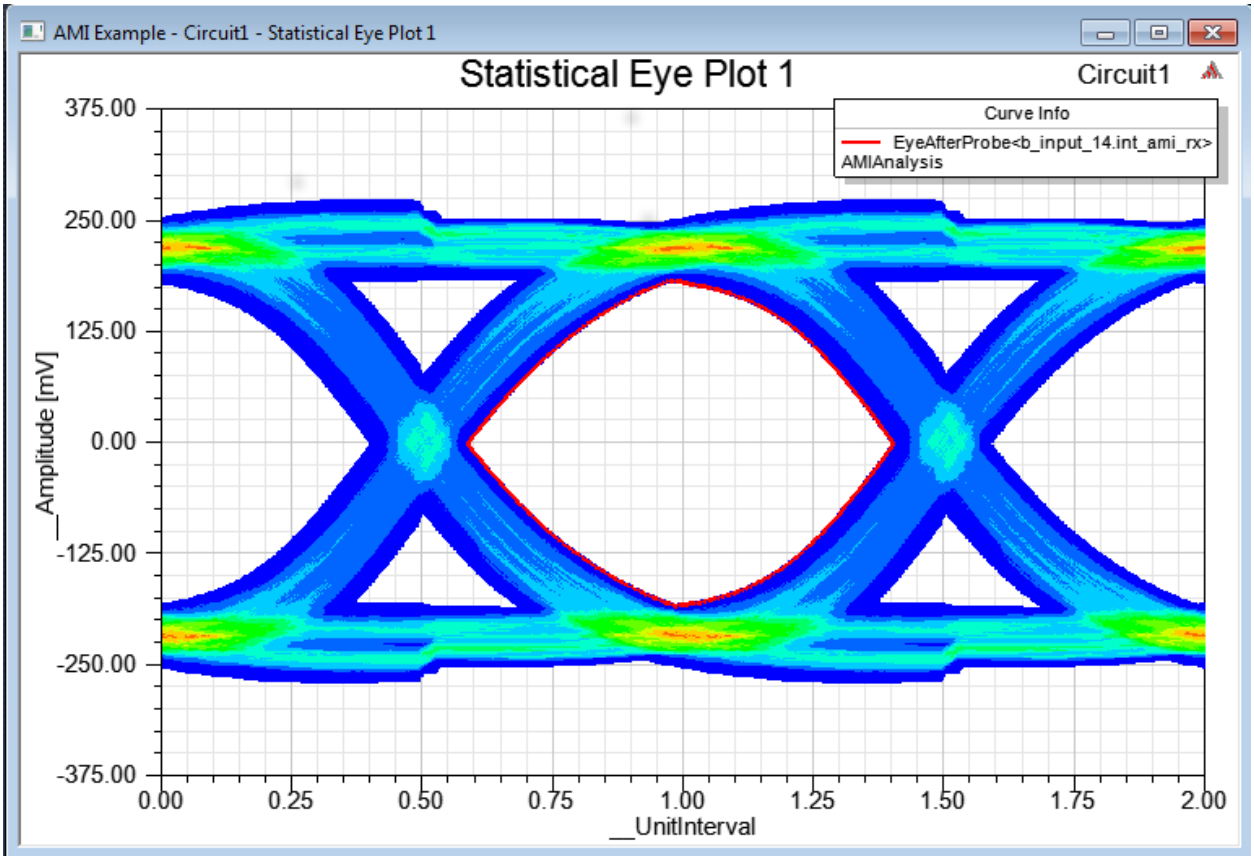
A Statistical Eye Plot resembles the following:



This eye diagram matches the UI extents displayed by the bathtub plot for the same trace and have a very similar eye opening position. The trace can be selected and the display changed to a "Front Panel" eye *(the default option for Statistical Eye Plots is a non-"Front Panel" eye)* via the "Eye" tab of the property window. This displays the [0, 2] UI extent front group box eye.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





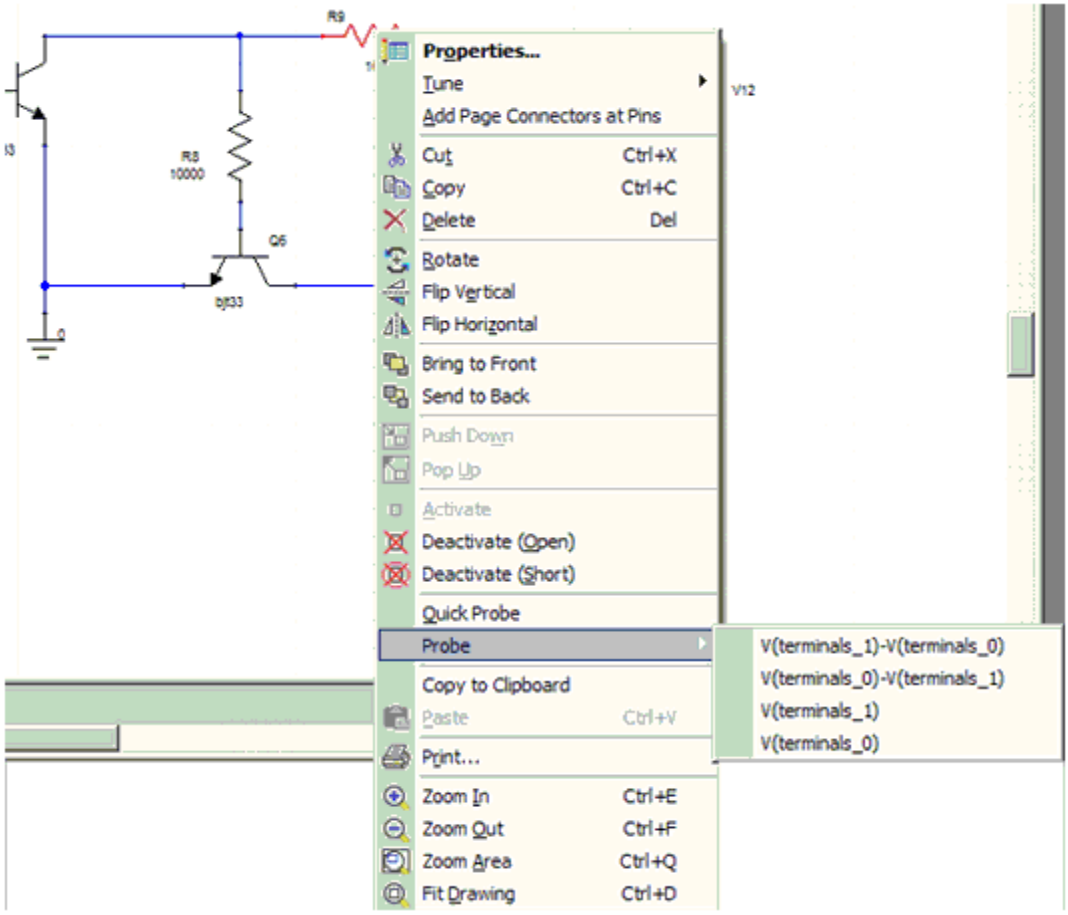
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D and Circuit Plot-On-Schematic

One convenient way to generate a 2D report is to create a **Plot-On-Schematic**. A Plot-On-Schematic generates a standard 2D report but also associates it with the component being simulated. It then displays the plot alongside the component in the **Schematic** editor.

After simulating an analysis, select a component in the **Schematic Editor**, right-click over the component and highlight **Probe**. When you highlight the **Probe** menu item, a list of terminals that can be plotted is displayed (such as voltage across the component or various combinations of terminals). For larger pin components, more options appear in the **Probe** list, such as voltage across various terminals (including collector-to-base, base-to-collector, individual terminals, and any current or power options).

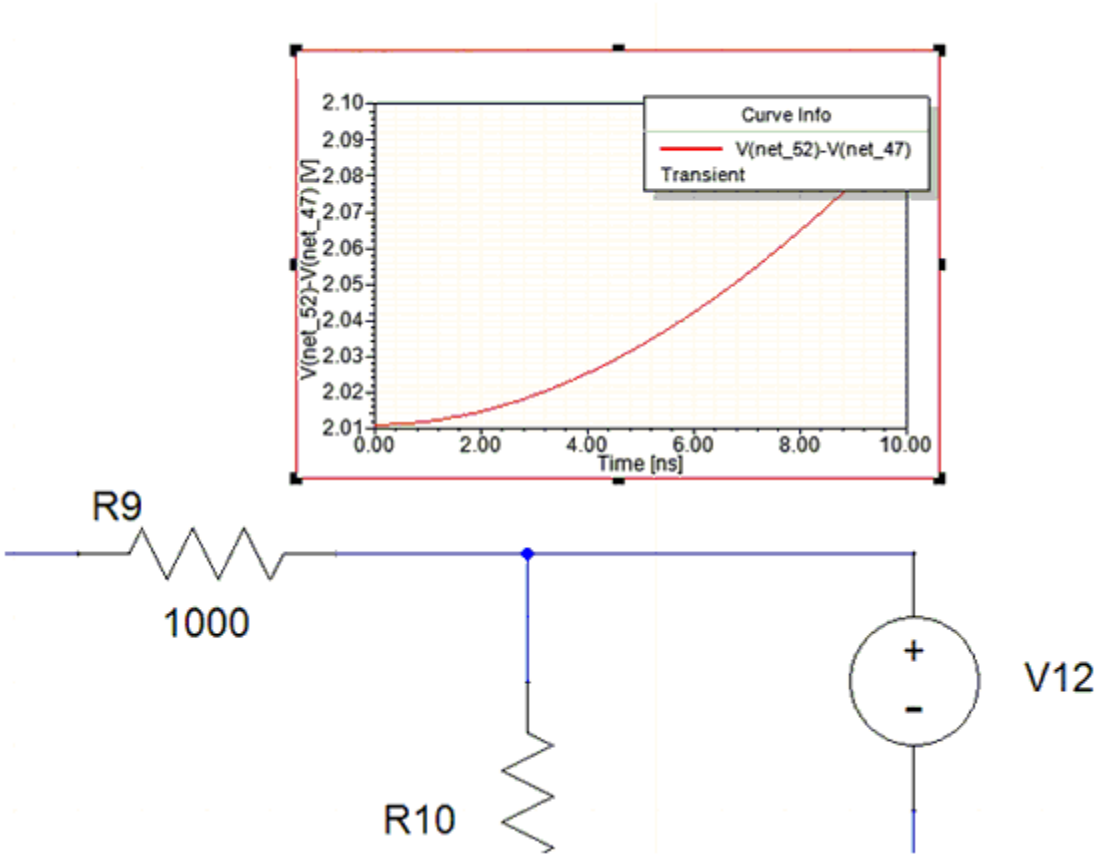
You can also generate a plot of a selected net, or a plot of multiple-selected nets. For instance, when you select two nets, a right-click option of **Quick Probe** allows you to create a **Plot-On-Schematic** of V(firstNet)-V(secondNet).



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 select an entry in the **Probe** list, the quantity is automatically plotted and placed on the schematic next to the selected component. The plot is associated with the component — as the component moves, the plots moves with it. (The plot can also be directly relocated by dragging and dropping.)

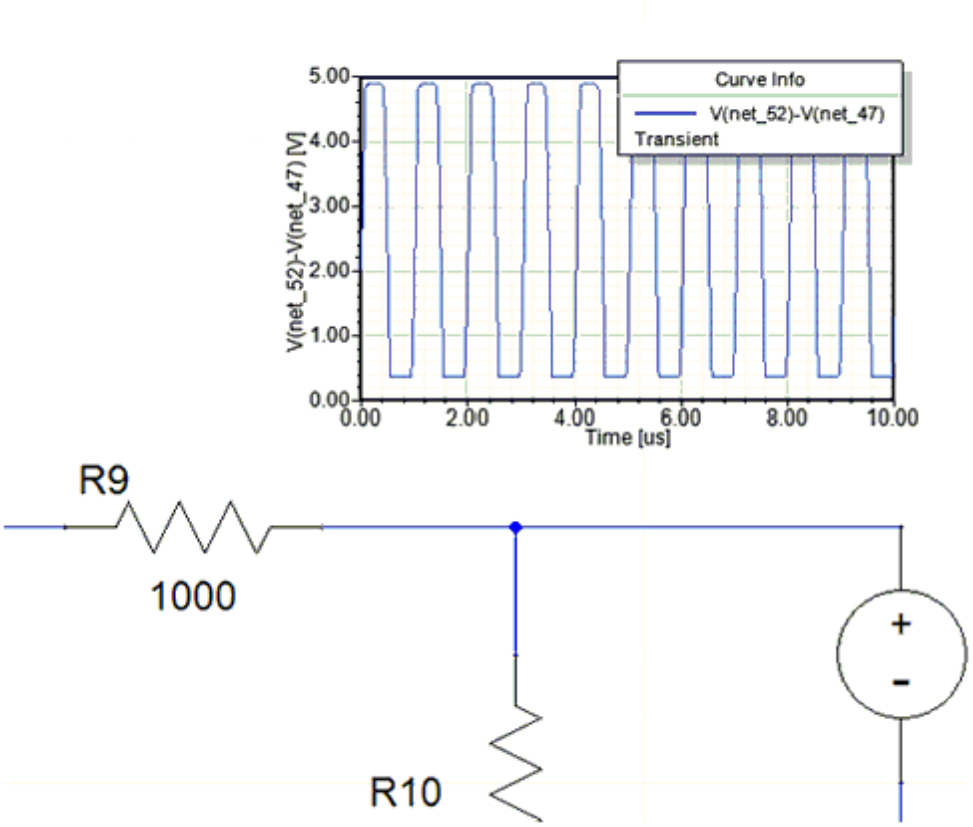
- The created plot also shows up in the **Results** folder in the **Project Tree**.
- Each component/net/port can have only one plot associated with it.
- You can have multiple plots on the same schematic, but only one such plot for each component.



If you right-click a plot in the schematic, choose the **Edit in Place** option (same as double-

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 plot). The plot then becomes fully editable.



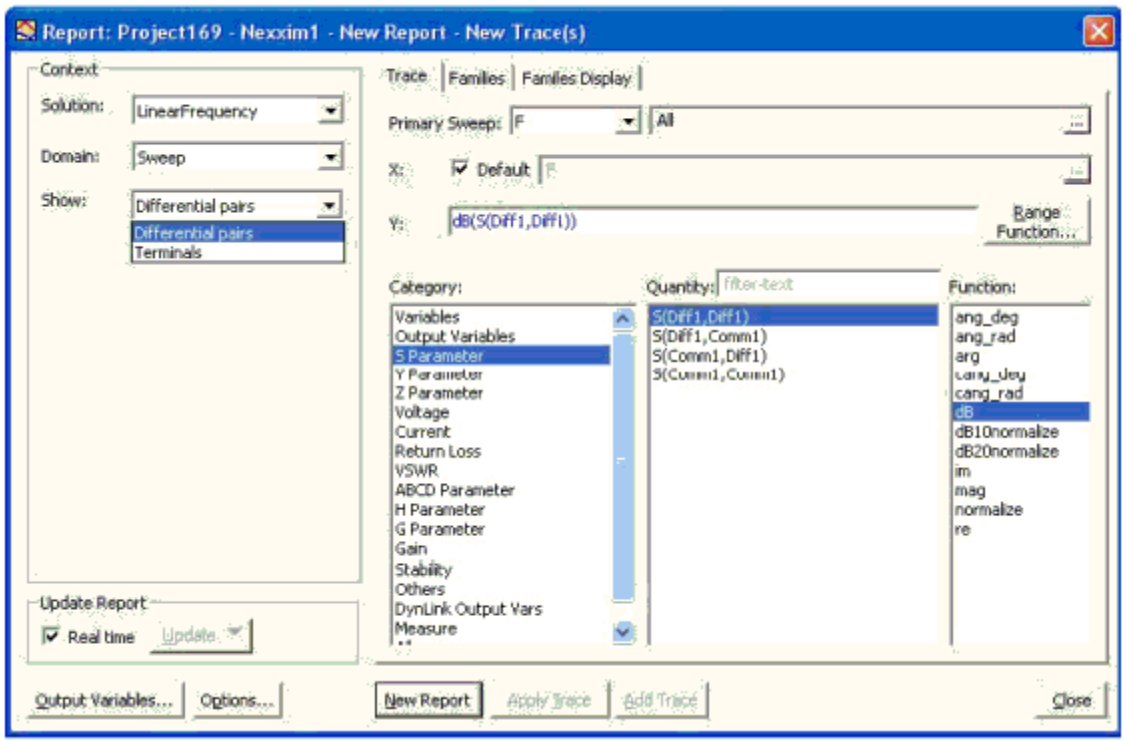
- You can move the legend bar around, change the color of traces, or change scaling for X and Y-axis — every operation that is normally performed in a regular plot window.
- You can also add plot markers.
- When you “click out” of the plot, it reverts back to being another schematic object.
- Plots are dynamically updated when you modify the analysis, so after you re-analyze, the **Plot-On-Schematic** is also updated.

For more information, see Modifying Reports.

2D and Circuit Reports with Differential Pairs

If differential pairs exist and are enabled, they change the reporter window – note the **Show** option in the **Context** group box. When differential pairs is selected as a **Show** option, the **Quantity** column changes on the original port names to the differential and common mode quantities.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2D and Circuit Creating a Report from a File

A report can be saved from one design and imported into another (or imported into the same design later). This allows you to take a static snapshot of a set of simulation data and view it at a later stage of the design process, or view the same dataset in one or more different designs.

To save a report to an Ansys Report Data File:

1. Create the report that you wish to save.
2. Right-click in the report window and select **Export** to open the **ExportReport** window.
3. Select the **Ansys ReportData files** option (.rdat) in the **Save as type** selection window.
4. Click the **Export Report** window to navigate to a appropriate location and either create a file name or select an existing file for the exported report. Then click **Save**.

To create a report from an existing Ansys Report Data File:

1. Right-click the **Results** folder in the **Project Manager** window and select **Create Report From File**.
2. Browse to select the appropriate **Ansys ReportData file** (.rdat). Then click **Open**.
3. The selected report is created in a new report 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:

- Most report types supported by Electronics Desktop (including Rectangular Plots, Smith Charts, and Eye Diagrams) can be exported to **Ansys ReportDatafiles**. However, three-dimensional plots can not be exported or imported.
- Limited editing of reports created from files is supported. For example, change the color of a trace or an eye mask definition.

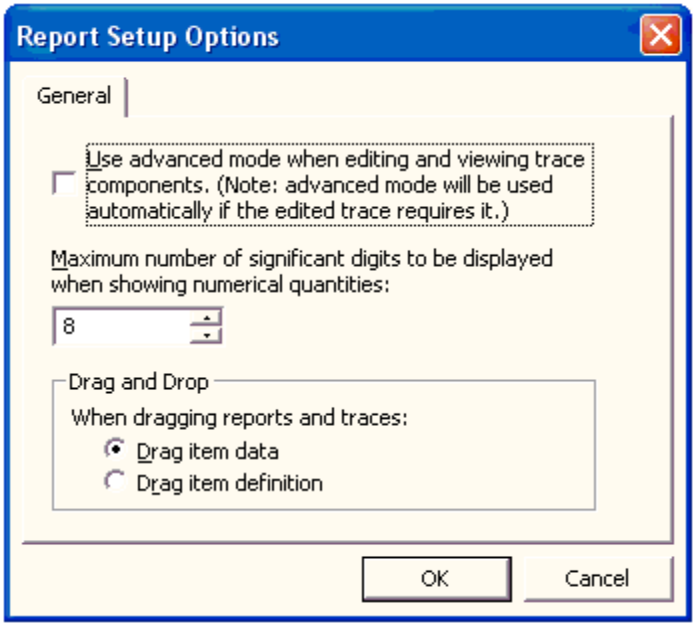
Setting 2D and Circuit Report Options

The following topics describe how to set up report options.

Setting Report Setup Options (2D and Circuit)

To set Report Setup Options in Electronics Desktop:

1. Click **Tools > Options > Report Setup Options** to open the **Report Setup Options** window.



2. Click the check box to specify whether to use **advancedmode** when editing and viewing trace components. (Advanced mode is used automatically if the trace requires it.)
3. Use the text-box/drop-down-menu to specify the number of **significantdigits** to use when displaying numeric values.
4. Specify the drag and drop behavior by clicking **Drag item data** or **Drag item definition**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Report2D Options

To set Report2D options in Electronics Desktop:

1. Click **Tools > Options > Report2D Options**. The **Report2D Options** window opens, displaying several available tabs.

   For properties controlled by check boxes, you can set values for all curves by clicking the column header cell that contains the property title. To change cell values, right-click on a cell field and select **Cut**, **Copy**, or **Paste**. To make changes to entire rows, right-click on a menu cell to select **Copy** or **Paste**.

   You can use a Restore Defaults button.

2. Click each tab, and make the appropriate selections.
3. Click **OK**.

Report2D Options: Curve Tab

These options are set on the **Curve** tab of the **Report2D Options** window.

1. Line style — select the options from the drop-down menu. The options are Solid, Dot, Dash, and Dot dash.
2. Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
3. Width — set the line width by editing the real value in the field.
4. Arrows — click the check box to use arrows on the curve ends.
5. Symbol — click the check box to have symbols mark the locations of data points on the curve.
6. Sym Freq — set the symbol frequency by editing the integer value in the field.
7. Sym Style — select the symbol to display for the designated data points. The sym style can be box, circle, vertical ellipse, horizontal ellipse, vertical up triangle, vertical down triangle, horizontal left triangle, horizontal right triangle.
8. Fill Sym — click the check box to set the symbol display as a solid or as hollow.
9. Sym Color — set the color for the symbol by double-clicking to open the Set color window. Select a default or custom color and click OK.

Report2D Options: Axis Tab

These options are set on the **Axis** tab of the **Report2D Options** window.

1. Axis Name — this describes the axis to which the following options refer.
2. Color — set the color by double-clicking to open the Set color window. 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.

3. Auto Scale — click the check box to toggle whether to auto scale the axis.
4. Min Scale — if Auto Scale is not selected, edit the real value to set the minimum value of the axis.
5. Max Scale — if Auto Scale is not selected, edit the real value to set the maximum value of the axis.
6. Auto Units — click the check box compute the correct units for the axis.
7. Units — click the cell to select from a menu of available units if you have not checked Auto Units.
8. Font color — set the font color of the axis by double-clicking to open the Set color window. Select a default or custom color and click OK.
9. Edit Font — click the cell to display the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.

Report2D Options: Grid Tab

These options are set on the **Grid** tab of the **Report2D Options** window.

1. Grid Name — lists the name or letter of the grid. Not editable.
2. Line Style — select the options on the dropdown menu. The options are Solid, Dot, Dash, and Dot dash.
3. Line Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.

Report2D Options: Header Tab

These options are set on the **Header** tab of the **Report2D Options** window. For the Title and subtitle, independently specify the following:

1. Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
2. Font — click the cell to open the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.

Report2D Options: Note Tab

These options are set on the **Note** tab of the **Report2D Options** window.

1. Note Color — set the color by double-clicking to open the Set color window. 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.

2. Note Font — click the cell to display the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.
3. Background Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
4. Background Visibility — click the check box to toggle the background for the note on or off.
5. Border Line Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
6. Border Visibility — click the check box to toggle the visibility of the note border.
7. Border Line Width — set the line width by editing the real value in the field.

Report2D Options: Legend Tab

These options are set on the **Legend** tab of the **Report2D Options** window.

1. Show Trace Name — click the check box to toggle the visibility of the trace name.
2. Show Solution Name — click the check box to toggle the visibility of the solution name.
3. Show Variation Key — click the check box to toggle the visibility of the variation key.
4. Text Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
5. Text Font — click the cell to display the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.
6. Background Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
7. Border Line Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
8. Border Line Width — set the line width by editing the real value in the field.
9. Grid Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.

Report2D Options: Marker Tab

These options are set on the **Marker** tab of the **Report2D Options** window.

1. Marker Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
2. Marker Font — click the cell to display the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

window.

3. X Marker — use the following options to set the X Marker properties.
    a. Show Intersection — check box to show the intersection.
    b. XMarker Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
    c. XMarker Font — click the cell to display the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.
    d. Box Background Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
    e. Line Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
    f. Line Style — select the options from the drop-down menu. The options are Solid, Dot, Dash, and Dot dash.
    g. Line Width — set the line width by editing the real value in the field.

Report2D Options: Marker Table Tab

These options are set on the **Marker Table** tab of the **Report2D Options** window.

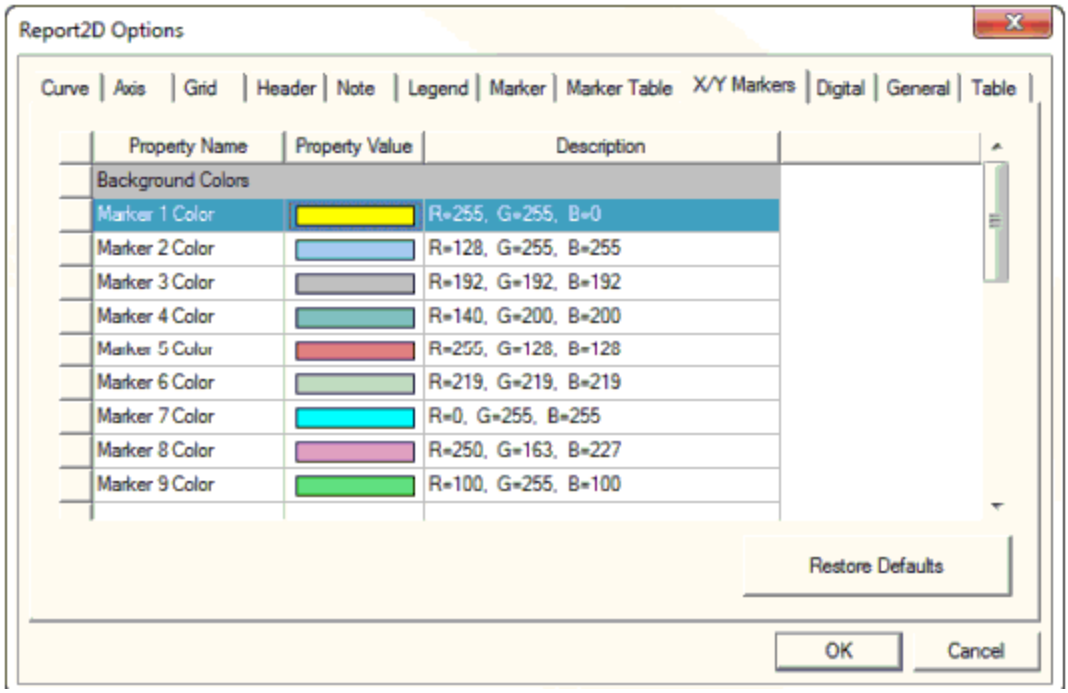
1. Precision — set the precision for marker placement by editing the real **Value** field.
2. Text Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
3. Text Font — click the cell to display the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.
4. Background Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
5. Border Line Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
6. Border Line Width — set the line width by editing the real value in the field.
7. Grid Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
8. Grid Line Width — set the line width by editing the real value in the field.

Report2D Options: X/Y Markers Tab

Use this tab on the **Report2D Options** window to set the properties for the markers.

Background colors for Markers 1 through 10. You can set these by select the current color to open a color selection window, or by specifying RGB number values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Properties, including

- On-screen intersection
- Marker Font
- Text color
- Line color
- Line style
- Line width
- Whether to Show Name
- Whether to Snap to Vertex

Inter marker deltas, including

- Whether to show delta
- Delta font
- Delta text color
- Line color
- Line style
- Line width

You can also **Restore Defaults**.

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: Digital Tab

These options are set on the **Digital** tab of the **Report2D Options** window.

Digital Literal Foreground color.

Whether to Expand Arrays/Records

Digital Stack Hight in Pixels for the following:

- Analog
- Digital
- Enum
- Event
- Literal.

You can also **Restore Defaults**.

Report2D Options: General Tab

These options are set on the **General** tab of the **Report2D Options** window.

1. Background Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
2. Contrast Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
3. Highlight Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
4. Accumulate Depth — sets the maximum number of accumulated traces on a single report. When Accumulate Depth is turned on (i.e., set to a non-zero value),run an analysis to generate a plot, then change some parameter value(s) and run another analysis. The newly generated plot now contains both old and new results that can then be compared. The plot keeps adding new results with each subsequent analysis until the accumulate length is reached.
5. Curve Tooltip Option — click the check boxes to toggle the following properties:
    a. Show Trace Name
    b. Show Variation Key
    c. Show Solution Name
6. Clipboard Option - specify the following properties:
    a. Capture Aspect Size Ratio — this can be As Shown or Full Screen.
    b. Capture Background Color — this can be As Shown or White.

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: Table Tab

These options are set on the **Table** tab of the **Report2D Options** window.

1. Text Font — click the cell to display the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.
2. Format — use the following properties to set the format:
    a. Field Width — set the table field width by editing the real value in the field.
    b. Precision — set the table precision by editing the real value in the field.
    c. Use Scientific Notation — click the check box to toggle scientific notation on or off.
3. Copy to Clipboard — use the following check boxes to toggle the following properties for table copy operations.:
    a. With Header
    b. With Tab Separator

Modifying 2D and Circuit Reports

After creating a report, See the following topics to modify the report's content, appearance and attributes.

Modifying 2D and Circuit Report Data

To modify the data that is plotted in a report:

1. From the **Project Manager** window, click the report you want to modify.
2. Right-click **Modify Report** to open the **Report**window.
3. The **Report** window command buttons permit you to create a new report with the settings you provide, or to modify an existing report.

- **Output Variables** – opens the **Output Variables** window.
- **Options** – opens the **Report Setup Options** window. 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.
- **New Report**. Adds a report to the Project tree under the Results icon. The new Report is displayed in **Electronics Desktop** window.
- **Apply Trace** – applies the selected traces in a report based on further processing or changes.
- **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
- **Close** – closes the **Report** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The updated report appears in the view window.

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*

Modifying the Background Properties of a 2D or Circuit Report

The standard View menu Zoom and Fit commands operate on reports. When zooming on a report, axis labels and ticks adjust automatically during the zoom operation and rescale to their final value after the zoom operation is complete.

To modify the appearance of a report, or the display properties an object in a report:

1. Open the report you want to modify.
2. You must select an editable object in the report to be able to edit its properties. Click an object to select it and view its Properties in the docked properties window. To open a floating Properties window, either double-click the selected object, or click **Edit > Properties** on the toolbar.

The selectable objects in reports are as follows:

- **Header** – this lets you edit 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 Manager** window, plot title synchronizes. The Company Name and the Show Design Name check box are grouped in the Properties window as Subtitle. Edits to the Subtitle Font Property affects both of them.
- **General —** this window (or General tab for other Report properties windows) lets you edit 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.)
- **Legend —** this lets you edit the Properties for whether to Show Trace Name, Solution Name, and Variation Key. At least one of these three must be selected. 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. Also see Modifying the Legend in a Report
- **Traces** – select traces either in the Legend or on the plot. The properties for traces include: 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. See Editing the Display Properties of Traces.
- X or Y **Axis Tab** — the defaults for most of these values are set in the **Report 2D Options** Axis tab.
  - Specify name — check box for specifying the Axis name.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Name — this describes the axis to which the following properties/options refer. These are selected in the **Report** window.
- Axis Color — set the color by double-clicking to open the Set color window. Select a default or custom color and click OK.
- Text Font — click the cell to open the Edit Text Font window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window.
- **Manual Format** (group box)

- Number format — choose a format (i.e., Auto, Decimal, or Scientific notation).
- Field Width — enter a real value.
- Field Precision — enter a real value.
  - X or Y **Scaling Tab** — These properties provide control over scaling.
- Axis Scaling — choose scaling as Linear or Log. For the Y axis, all zero or negative values are discarded before log scaling is applied.
- Specify Min — check box
- Min — text entry in same units as axis units. Saved as SI internally.
- Specify Max — check box
- Max — text entry in same units as axis units. Saved as SI internally.
- Specify Spacing — check box
- Spacing — text entry in same units as axis units. Saved as SI internally
- **Manual Units** (group box)
- Auto Units — click the check box compute the correct units for the axis.
- Units — click the cell to select from a menu of available units if you have not checked Auto Units.
- **Infinity Visualization** (group box)
- Map Infinity Mode — check box.

Each axis now 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 Electronics Desktop treats as zero, in the data, for example, dB Gain.

- Map Infinity To — enter a real value for the Map Infinity Mode.

3. Edit the properties, and **OK** the window to apply 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.

Modifying the Legend in a 2D or Circuit Report

The legend in a report is a list of the curves being plotted. For each curve, the legend gives the name, shows the line color, and lists the setup and the adaptive pass used to generate the curve.

To show or hide a legend in a report:

1. Make the report the active view.
2. Use **View > Active View Visibility** or the **Show/Hide** icons on the toolbar to display or hide the report.

   Either command displays the **Active View** window.

3. Select the **Legends** tab.

   This lists the legend (or legends) in the report.

4. Check the visibility check box, and OK the window to close it and apply the change.

To edit the display properties of a legend:

1. Select the legend in a report by clicking on the group box to display a docked properties window, or right-click on the legend and select **Edit > Properties** to open the floating properties window.

   This lets you edit the Properties for whether to Show Trace Name, Solution Name, and Variation Key. At least one of these three must be selected.

   You can also edit the Font by clicking the Font cell to open the **Edit Text Font** window. The window lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The window also contains a preview field. OK the selections to apply the font edits and close the window

   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 window and apply the selections. To **change the display name** for traces, see Editing Trace Properties.

To move a legend in a report:

1. Click and hold and 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Position the mouse tip over the edge you want to resize.

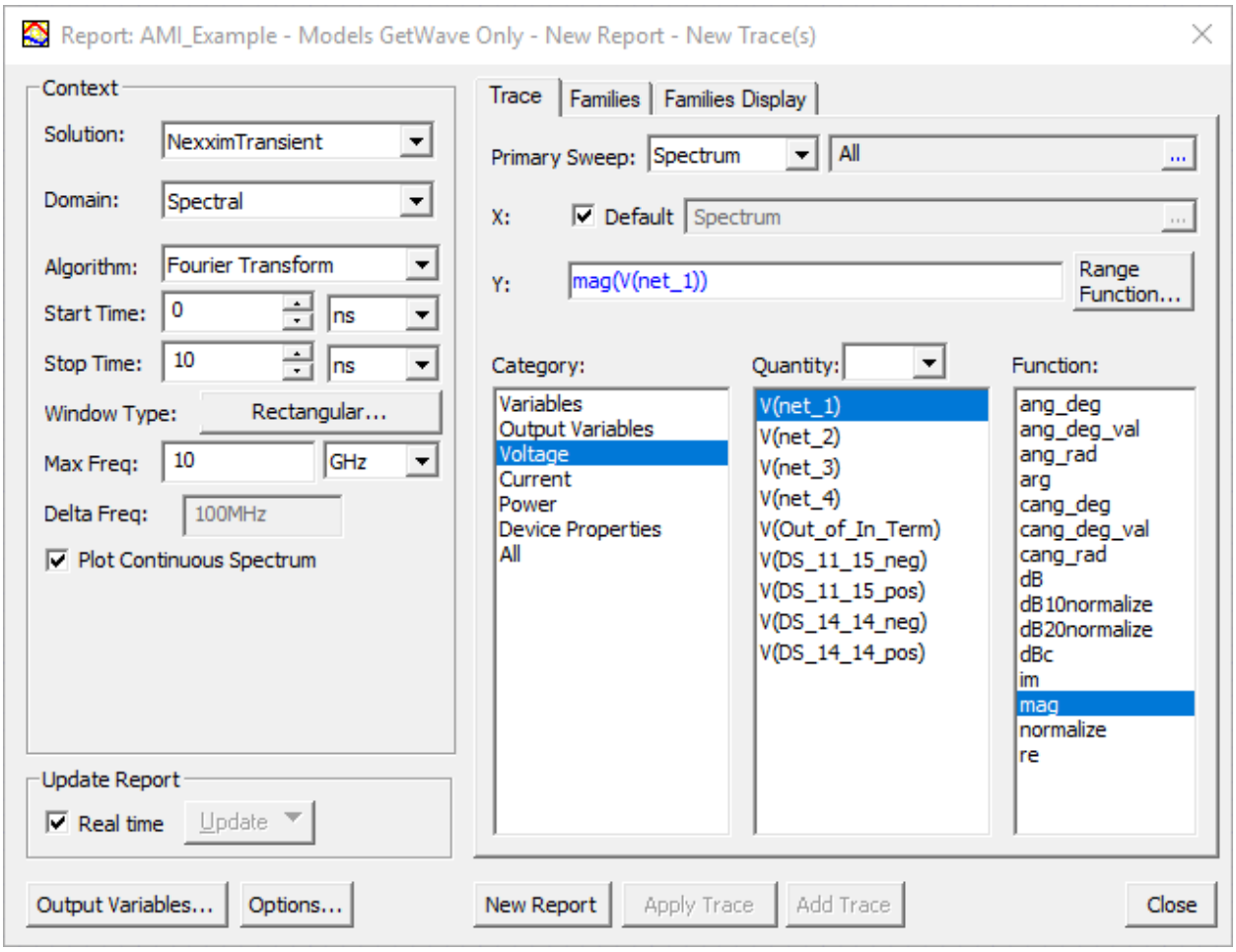
The mouse pointer changes to a horizontal or vertical line with arrow tips.

2. **Click+drag** the horizontal or vertical edge to the desire size.
3. Release.

Plotting Circuit Spectral Domain Data

After transient analysis, one can exam frequency components of the signal of interest by plotting spectral domain data on the Reporter.

1. Open a project in **Project Manager** window.
2. Right-click **Results**.
3. In **Solution**, choose **NexximTransient**.
4. In **Domain**, choose **Spectral**. Several fields appear in the **Report** window:



In the **Context** pane:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Start Time** and **Stop Time** define the signal duration T_0 = **Stop Time** - **Start Time**.

  > **Note: Stop Time** and **Start Time** should lie inside the transient simulation time limits.

- **Max Freq** sets the maximum frequency for spectral analysis and plotting.
- **Delta Freq** is automatically supplied as $1/T_0$ Hz.
- The **Algorithm** drop-down menu contains 3 options: **Fourier Integration** (FI), **FFT**, and **Fourier Transform**.

  Fourier Integration and **FFT** both provide Fourier Series (or Fourier Coefficients) of the transient signal with DFT, where periodicity is implicitly assumed and the period is defined as T_0 = **Stop Time** - **Start Time**.

  Fourier Integration computes Fourier series by numerically integrating the input waveform from its transient simulation. The sampled waveform may not be on a set of evenly spaced time points. **Fourier Integration** is much slower compared with **FFT**, but it can be more accurate since it preserves the shape of underlying waveform without re-sampling.

  The **FFT** algorithm re-samples the input waveform by interpolating to evenly spaced time points. It uses the FFTW library that is publicly available and whose computational complexity is

  $O(Nlog_2N)$

  where N is the sample size after re-sampling and N = 2 *#Harmonics +1, where #Harmonics = **Max Freq** /**Delta Freq**.

  > **Note:**
  >
  > - Based on the Nyquist-Shannon Theorem or Sampling Theorem, the sampling rate should be at least twice the bandwidth of the signal to avoid aliasing.
  > - Spectral plots based on **Fourier Integration** and **FFT** are single-sided with magnitudes of non-zero harmonics doubled because the negative frequency components are folded to the positive side.

  Fourier Transform (FT) provides the Fourier transform of a signal defined in [**Start Time**, **Stop Time**] by leveraging the **FFT** algorithm. The Fourier transform of a time limited signal f(t) in [0, T_0] is

  $$F(f) = \int_{-\infty}^{\infty} f(t)e^{-i2\pi ft}dt = \int_{0}^{T_0} f(t)e^{-i2\pi ft}dt$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Fourier series of a periodic signal that is equal to f(t) in [0, T_0] and with period T_0 is

$$c_n = \frac{1}{T_0}\int_0^{T_0} f(t)e^{-i\frac{2\pi}{T_0}nt}dt$$

Hence,

$$c_n = \frac{1}{T_0}\mathrm{F}\left(\frac{n}{T_0}\right)$$

In other words, the Fourier series of f(t) based on **FFT** can be multiplied by T_0 in order to get the **Fourier Transform** of f(t) at sampled frequencies n/T_0.

> **Note:** Spectral plots based on **Fourier Transform** only show the positive frequency side.

- The **Plot Continuous Spectrum** check box provides a continuously plotted spectrum if selected and a discrete spectrum when cleared.
- **Window Type** determines which window type to apply.

  When applied to transient data, a window type can help resolve closely spaced harmonics and reduce spectral leakage due to the finite nature of the time signal. The following window types are available. (The “t” parameter is normalized time.)

  - **Rectangular**: 1.0
  - **Bartlett** (also known as the Parzen or triangular window): 1.0 - t
  - **Blackman**: 0.42 + 0.5*cos(π*t) + 0.08*cos(2*π*t)
  - **Hamming**: 0.54 + 0.46*cos(π*frac)
  - **Hanning**: 0.5 * (1.0 + cos(π*t))
  - **Kaiser**: BesselI0(KaiserParam * sqrt(1.0 - t*t)) / BesselI0(KaiserParam)
  - **Welch**: 1.0 - t^2
  - **Weber**: (0.828217*π^3*t^3 - 1.637363*π^2*t^2 + 0.041186*π*t + 0.99938)
  - **Lanzcos**: sin(π*t)

  Optionally, click the **Adjust Coherent Gain** check box to adjust signal levels (based on the window type) as if no window are used. When non-rectangular windows are used, the window automatically changes the power level of the signal. This is known as the “coherent gain” or “processing loss” of the window.

References:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[1] Fredric J. Harris, "From the Use of Windows for Harmonic Analysis with the Discrete Fourier Transform," *Proceedings of the IEEE* 66, no. 1 (Jan 1978): 51-83, http://web.mit.edu/xiphmont/Public/windows.pdf.

[2] Charan Langton and Victor Levin, "Fourier Transform of continuous and discrete signals," *The Intuitive Guide to Fourier Analysis and Spectral Estimation with MATLAB* (Mountcastle Academic, 2017), chap. 4, http://complextoreal.com/wp-content/uploads/2012/12/fft4.pdf.

[3] Paul Heckbert, "Fourier Transforms and the Fast Fourier Transform (FFT) Algorithm" (Notes 3 class handout for Computer Graphics 2, 15-463, Carnegie Mellon University, Pittsburgh, PA, Jan 27,1998), http://www.cs.cmu.edu/afs/andrew/scs/cs/15-463/2001/pub/www/notes/fourier/fourier.pdf.

[4] Rolf Isermann and Marco Münchhof, " Spectral Analysis Methods for Periodic and Non-Periodic Signals," *Identification of Dynamic Systems* (Springer, Berlin, Heidelberg, 2011), chap. 3, https://link.springer.com/chapter/10.1007%2F978-3-540-78879-9_3.

[5] Julius O. Smith III, *Mathematics of the Discrete Fourier Transform (DFT) with Audio Applications*, 2nd ed. (W3K Publishing, 2007), https://www.dsprelated.com/freebooks/mdft/.

Delta Markers in 2D Reports (2D and Circuit)

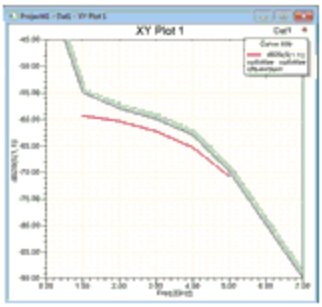
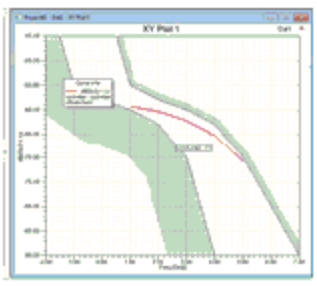
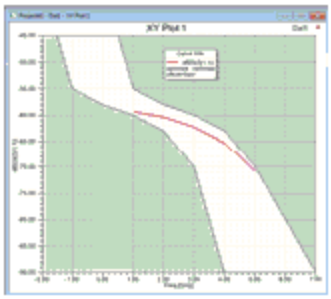
To view the difference between any two marker points in a report:

1. Set the first marker by clicking and holding the mouse button.
2. Move the mouse — without releasing the left — to another position, then release the left button to create the second marker.

The marker text window displays the difference between the two markers, instead of the absolute X and Y values.

Limit Lines in 2D and Circuit Cartesian Plots

Limit lines are simple graphical representations of constraints on XY plots. These are modeled as a sequence of XY point pairs. For example, designate a single limit line to delineate either an upper or lower limit, or two limit lines to delineate upper and lower limits. Add as many limit lines as you want. You can control the display properties of the line including color and hatch width.

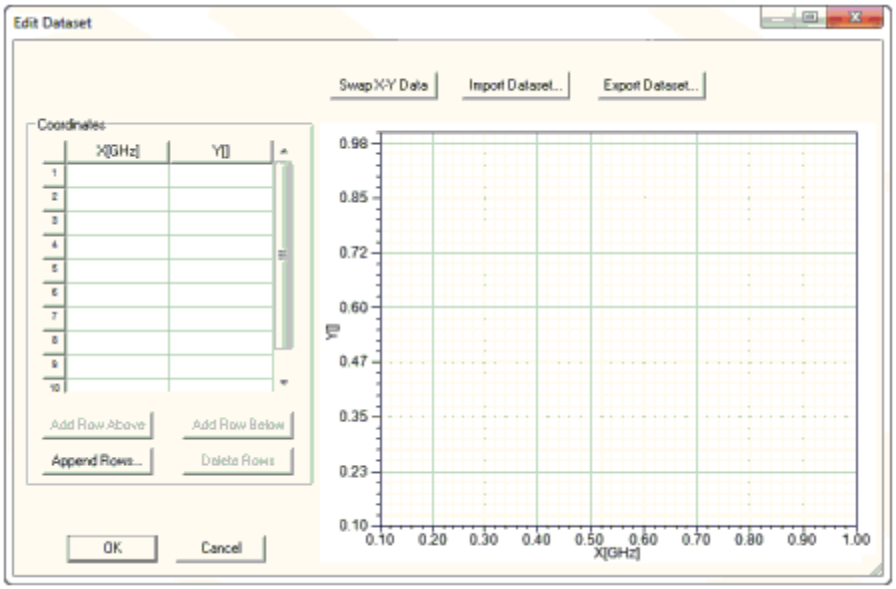
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Limit lines are available only on Rectangular (XY) plots, not on XY-like plots such as Bode or Rectangular Stacked.

To create a limit line:

1. Click **Report2D>Add Limit Line**, or right-click an XY plot and select **Add Limit Line**.

This opens an **Edit Dataset** window.



1. You can use this window to:

- Enter the XY coordinate values directly.
- Import XY values from a tab-delimited **.tab** file.
- Export dataset to a file.

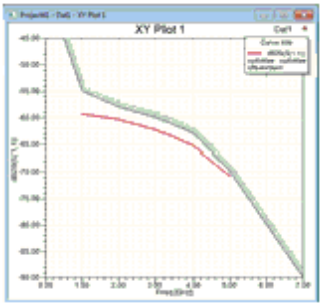
To add additional data points, click **Append Rows ...**, or modify the table by clicking **Add Row Above**, **Add Row Below**, and **Delete Rows**, as necessary.

You can press **Shift** +click to select multiple adjacent rows, or **Ctrl**+click to select any combination of rows for deletion.

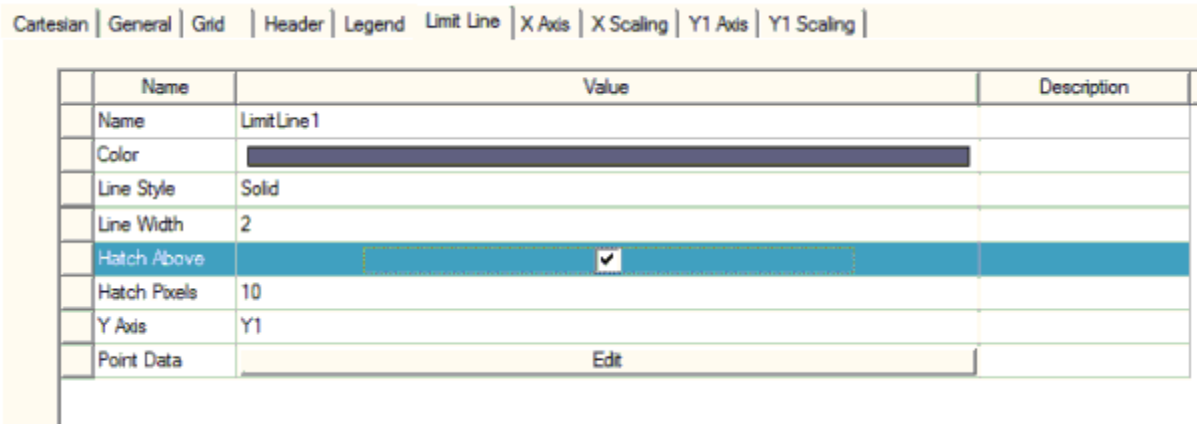
Note: Each limit line is associated with a particular Y axis (because it has to be scaled the same way as all the curves associated with the axis, follow its log/linear scale et cetera). This Y axis association defaults to the first available Y axis when the limit line is created. However, if the plot contains multiple Y axes, it can be associated with a different Y axis later via its properties tab.

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 **OK** to add the limit line you defined to the plot. The line divides the plot into two regions in the context of its length. By default, the upper region is hatched to designate constraint violation.



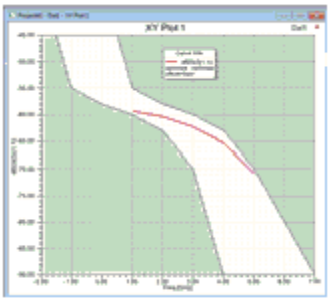
1. You can select the limit line in the plot to edit its properties on the **Limit Line** tab of the plot properties.



- Line properties: color, style and width
- Hatch direction (hatch above or below the limit line)
- Hatch width in pixels
- Y axis association
- The point data that defines the limit line itself

2. If you add a second limit line, designate it as hatch below by unchecking **Hatch Above** to produce a “tunnel” marking the 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.



2D and Circuit Spinning a 3D Report

You can set a 3D report spinning around any axis, to enable viewing from multiple angles without having to rotate repeatedly.

1. From the **View** menu, select **Spin**. Alternatively, right-click the 3D graph, select **View** on the dropdown menu, and select **Spin** on the subordinate menu. The cursor changes to the Spin mode cursor (curved arrows).
2. **Click+drag** the mouse in the direction and at the speed you want to spin the view. When you release the mouse button, the image begins spinning continually.
3. To stop spinning, click the mouse again in the report window. Start another spin in the same or another direction and speed by repeating step 2.
4. To end Spin mode, press **Esc** or deselect **Spin** on the **View** menu.

Reviewing 2D Polar Plots

For a polar plot of S-parameters, Electronics Desktop displays in the lower-left corner the following derived information about the cursor's location:

MP	The magnitude and phase of the point.
RX	The normalized resistance (**R**) and reactance (**X**).
GB	An alternate view of the normalized resistance and reactance in the form of

$$R + jX = \frac{1}{G + jB}$$

where

- ***G*** = conductance
- ***B*** = susceptance

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VSWR

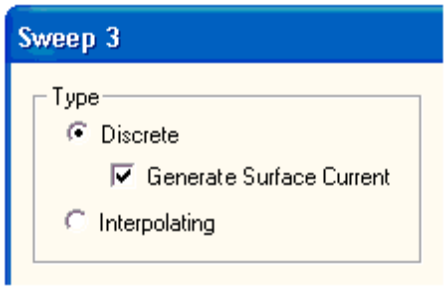
The voltage standing wave ratio, calculated on the equation $\frac{1+|S_{ij}|}{1-|S_{ij}|}$.

A scale under the plot displays the scale of points along the R-axis.

2D Overlaying Surface Currents on a 3D View

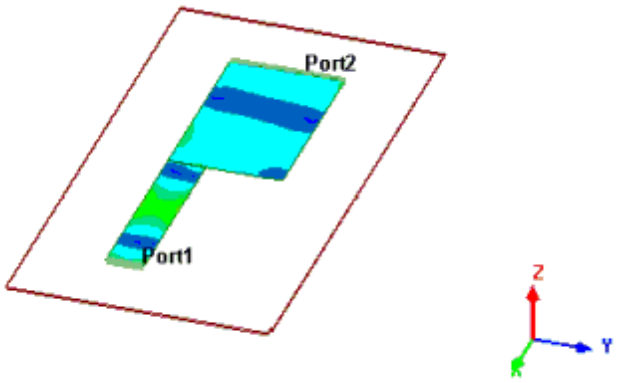
Surface currents calculated as the results of a simulation can be displayed as overlays on the 3D viewer.

1. To ensure that the surface current information is generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled (checked):

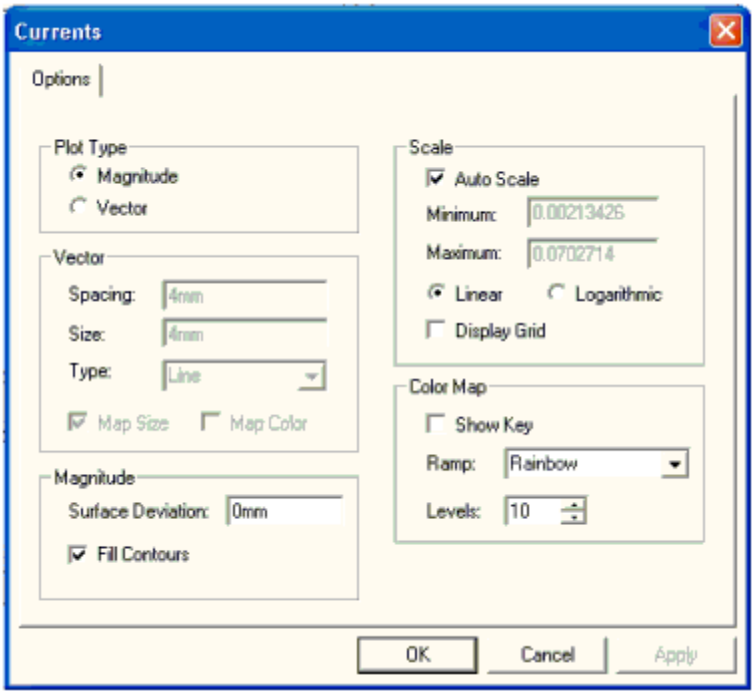


2. Run the analysis with the sweep.
3. to open the surface current as an overlay, expand the **Analysis** icon in the **Project Manager** window, and select **Setup***m* > **Sweep***n* > **Results** > **Display Currents** (*m* and *n* identify the particular solution setup and sweep setup, respectively). Also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project Tree**. The 3D viewer window opens with the current values overlaid on 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.



4. To change the display properties of the surface current overlay, expand the **Results** icon in the Project window, and select **Setup***m*:**Sweep***n*:**Currents***k* > **Properties** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively). The **Currents** window opens:



In the **Plot Type** group box, select **Magnitude** to enable the **Magnitude** group box options or select **Vector** to enable the **Vector** group box options.

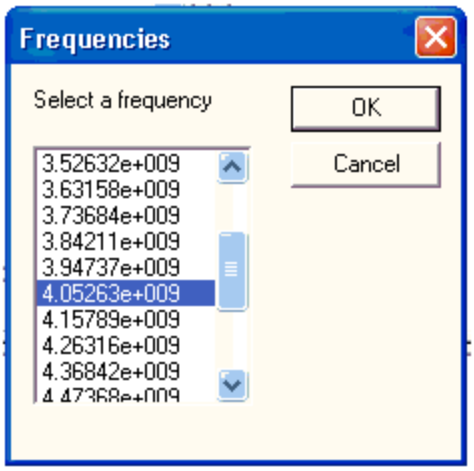
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Scale** group box, select **Auto Scale** (the default), or deselect **Auto Scale** and enter custom **Minimum** and **Maximum** scaling values. Select **Linear** or **Logarithmic** scaling (the default is **Linear**), and toggle **Display Grid** on or off (the default is off).

In the **Color Map** group box, select the **Ramp** type (**Rainbow** is the default; other options are **HueScale**, **Magenta**, and **Temperature**), set the number of **Levels** (the default is 10 levels), and toggle the color key (**Show Key**) on and off (default is off).

Click **Apply** to apply any changes to the display without closing the window. Click **OK** to apply any changes and close the window. Click **Cancel** to close the window without changing any options.

5. To select the frequency for the current overlay, expand the **Results** icon in the **Project Manager** window, and select **Setup***m*:**Sweep***n*:**Currents***k* > **Frequency** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively). The **Frequencies** window opens:



The list displays the frequencies that are swept in the analysis. When you select a frequency on the list, the overlay displays the surface current values calculated at that frequency. Click **OK** to leave the overlay at the selected frequency, or click **Cancel** to close the window without applying any frequency changes to the overlay.

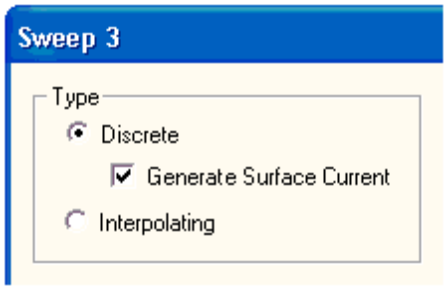
6. To dismiss the overlay, expand the **Results** icon in the **Project Manager** window, right-click **Setup***m*:**Sweep***n*:**Currents***k*, and select **Delete s** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively).

2D Overlaying Far Fields on a 3D View

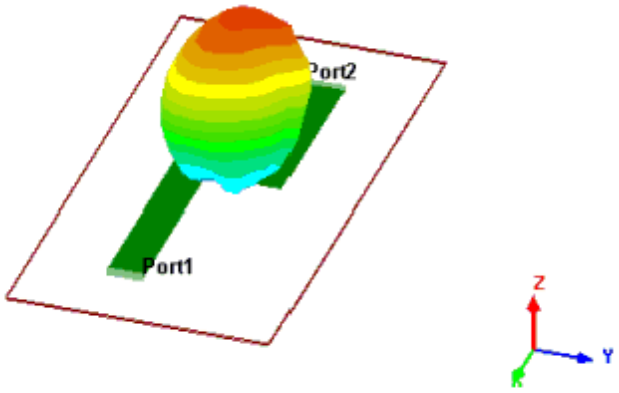
Far fields calculated as the results of a simulation can be displayed as overlays on the 3D viewer.

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 ensure that the far field information can be generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled (checked):



2. Run the analysis with the sweep.
3. to open the far field as an overlay, expand the **Analysis** icon in the **Project Manager** window, and select **Setup***m* > **Sweep***n* > **Results** > **Far Field** (*m* and *n* identify the particular solution setup and sweep setup, respectively). Also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project Tree**. The 3D viewer window opens with the far field values overlaid on the geometry:

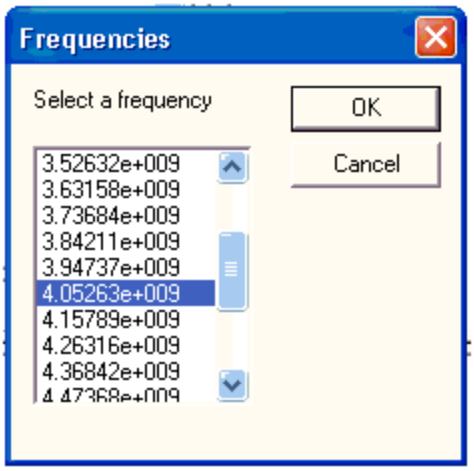


The display properties of the Far Field overlay cannot be changed. The Ramp type is Rainbow, and the number of levels is 20.

4. To select the frequency for the far field overlay, expand the **Results** icon in the **Project Manager** window, and select **Setup***m*:**Sweep***n*:**Far Field***k* > **Frequency**. The

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequencies window opens:



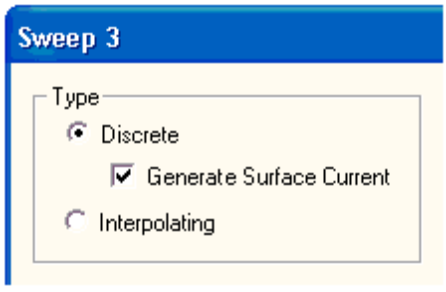
The list displays the frequencies that are swept in the analysis. When you select a frequency on the list, the overlay displays the far field values calculated at that frequency. Click **OK** to leave the overlay at the selected frequency, or click **Cancel** to close the window without applying any frequency changes to the overlay.

5. To dismiss the overlay, expand the **Results** icon in the **Project Manager** window, right-click **Setup*m*:Sweep*n*:Far Field*k***, and select **Delete** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and far field setup, respectively).

2D Overlaying Near Fields on a 3D View

Near fields calculated as the results of a simulation can be displayed as overlays on the 3D viewer.

1. To ensure that the near field information can be generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled (checked):



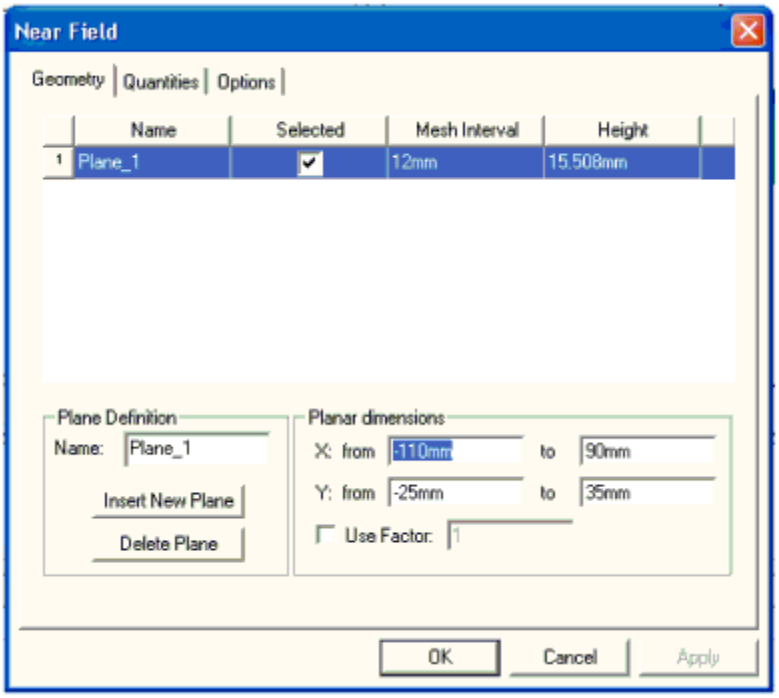
2. Run the analysis with the sweep.

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 open the near field overlay, expand the **Analysis** icon in the **Project Manager** window, and select **Setup***m* > **Sweep***n* > **Results** > **Near Field** (*m* and *n* identify the particular solution setup and sweep setup, respectively). Also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project Tree** to open the **Near Field** window. The window has three tabs and at the bottom of each tab are three buttons:

- **Apply** is activated whenever you change a value. Click **Apply** to start the display, and see the effect of each change. The window stays open.
- When no values are changed on any tab, the **OK** button starts the display. When one or more values have been changed, **OK** applies the changes. In either case, **OK** closes the window and adds an icon for the overlay under the **Results** icon in the Project window.
- **Cancel** is active as long as no changes have been applied. **Cancel** closes the window without changing any values. If the overlay is already displayed, it does not change. If **Cancel** is pressed before any overlay is displayed, the overlay is canceled.

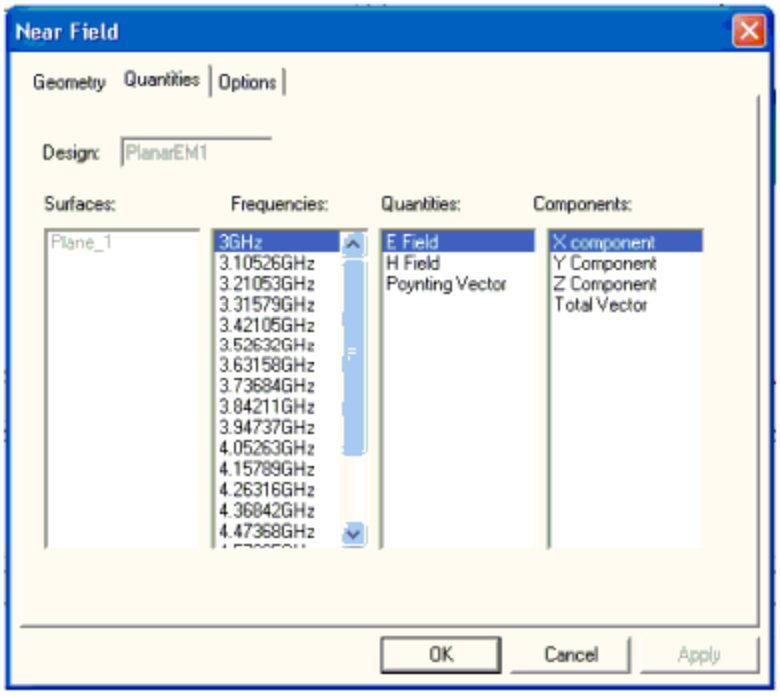
4. The **Near Field** window opens with the **Geometry** tab displayed:



5. Use the options in the **Geometry** tab to select (or define) one or more planes for the calculation, including the dimensions to be used, and a scale factor if appropriate.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

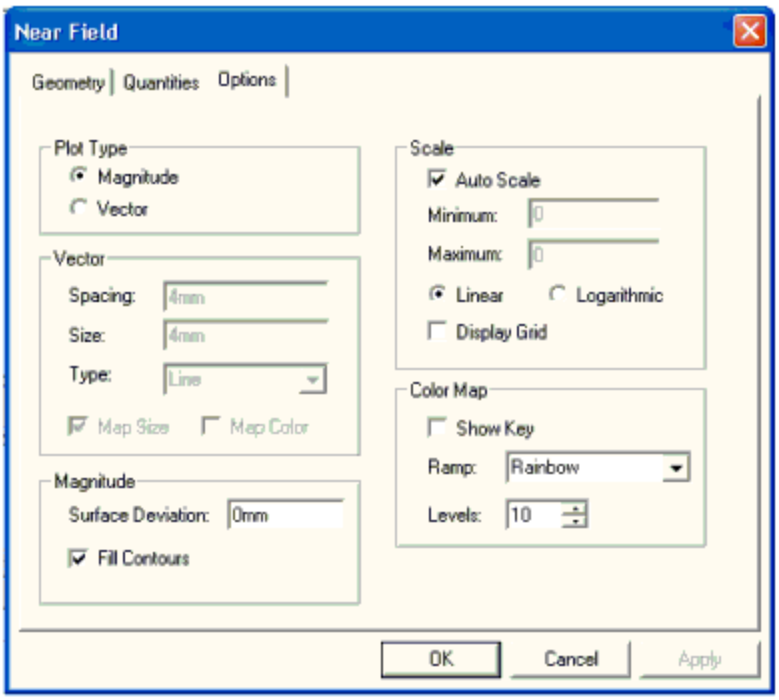
6. Click the **Quantities** tab to select the near field quantity to be calculated:



7. If more than one surface is involved, select a surface on the **Surfaces** group box.
8. Select a frequency on the **Frequencies** list. The frequencies are the ones swept in the analysis.
9. Select a field type on the **Quantities** list. **E** is the electronic field, **H** is the magnetic field, and the **Poynting Vector** is the (E×H*) field, where H* is the complex conjugate of the H matrix.
10. Select a vector component on the **Components** field.

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 the **Options** tab to specify the display options for the near field overlay:



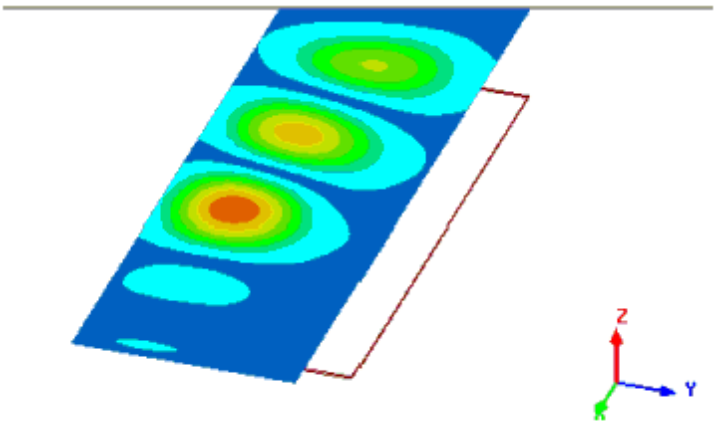
12. In the **Plot Type** group box, select **Magnitude** to enable the **Magnitude** group box options or select **Vector** to enable the **Vector** group box options.

13. In the **Scale** group box, select **Auto Scale** (the default), or deselect **Auto Scale** and enter custom **Minimum** and **Maximum** scaling values. Select **Linear** or **Logarithmic** scaling (the default is **Linear**), and toggle **Display Grid** on or off (the default is off).

14. In the **Color Map** group box, select the **Ramp** type (**Rainbow** is the default; other options are **HueScale**, **Magenta**, and **Temperature**), set the number of **Levels** (the default is 10 levels), and toggle the color key (**Show Key**) on and off (the default is off).

15. When you click **Apply** or **OK** in any of the **Near Field** window tabs, the 3D viewer window opens with the near field values overlaid on 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.



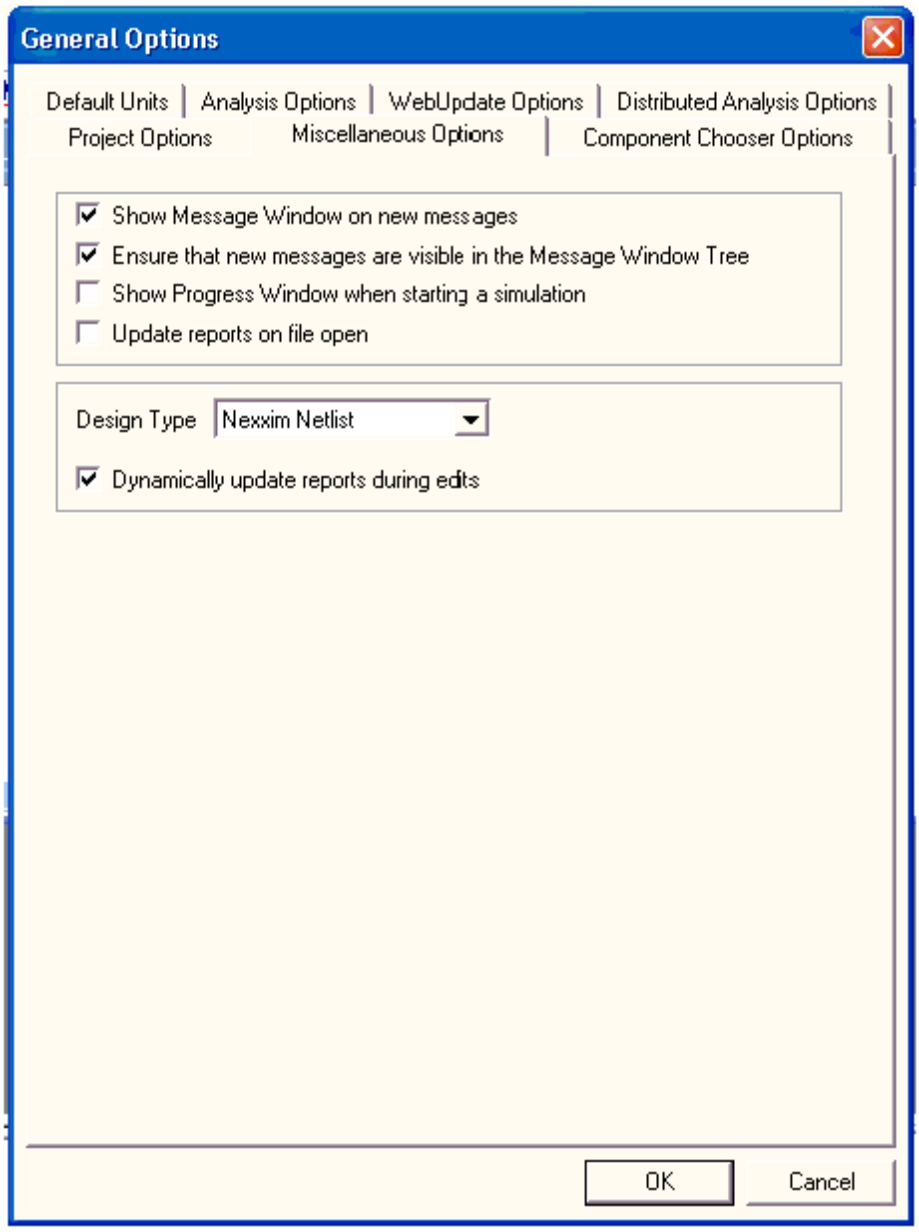
16. To dismiss the overlay, expand the **Results** icon in the **Project Manager** window, right-click **Setup** > **Sweep** > **Near Field**, and select **Delete**.

2D and Circuit Updating Post-processing Data

To manually update the report that is active in the design area using the latest simulation results, right-click in the report and select **Update Report** on the menu. Alternatively, click the **Report** entry in the menu and select **Update Report**(the text of the **Report** entry depends on the type of report being displayed).

In a schematic design, an option is available to control all schematic-based analyses. Select **Nexxim Circuit Options** from the **Tools > Options > General Options** drop-down. Here is the **Nexxim 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.



On all three options windows, the option **Dynamically update reports during edits** is available beneath the **Miscellaneous Options** tab and controls the version of solution data to be reported.

Electronics Desktop solution data is versioned. You might have solution data for a particular state of the Design, then edit the Design and re-simulate. Solution data is now available for both states of the Design. Suppose the following sequence occurs:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You simulate and create a report. The report displays the current solution data (state 1).
- You edit the design. The report is marked 'Invalid' (a large "X" appears in the upper-right corner of the report window and on the report icon in the Results folder in the **Project Manager** window.
- You rerun the simulation. The report is updated with the latest solution data (state 2), and the report is marked as valid (the "X" disappears).
- Now you click **Undo**. Electronics Desktop still has the 'old' solution data (state 1). If the option **Dynamically update post-process data on edit** is off, the State 2 report data becomes invalid again. If **Dynamically update post-process data on edit** is on, Electronics Desktop reloads the now valid state 1 solution data into the plot and marks it valid.
- Upon **Redo**, the result again depends on the setting of the **Dynamically update post-process data on edit** option. If the option is off, the report becomes valid again (the state 1 data was not reloaded by the **Undo** operation, so the report is still displaying state 2). If the option is on, the state 2 data is reloaded dynamically and plot is also valid.

Deleting 2D and Circuit Reports

To Delete a Single Report

To delete a single report, right-click the report in the **Results** folder in the **Project Manager** window for the design, and click **Delete** on the menu. Or, select a report icon in the **Project Tree**, and right-click to open the shortcut menu, then click **Delete** on the shortcut menu or the "**X**" icon on the toolbar to delete the selected report.

To Delete All Reports

To delete all reports, on the **Product** menu, select **Results** > **Delete All Reports**, or, on the **Project Manager** window, expand the **Project Tree**. Then right-click the [*active design folder*] and select **Delete All Reports**. You can also right-click **Results** and select **Delete All Reports**.

Working with Traces in 2D and Circuit Reports

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 one or more variables has been solved, at several values, "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 four independent y-axes. Traces appear in the **Project Manager** window 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 a selection can 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.

- For Windows 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.

Adding a Trace to a 2D or Circuit Report

In general, to add a trace to a report:

1. Select a report in the **Project Manager** window and right-click and select **Modify Report.**
2. In the **Report** window specify the Y component information.
   a. From the drop-down menu, select the Category of information you want to plot.

      The Category drop-down menu lists the available categories for the Solution type and the active 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 on 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. 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 fields. Invalid functions appear in red.
   e. **Range Function** — opens the **Set Range Function** window. This applies currently specified Quantity and Function.
3. In the **Report** window specify the X axis information (e.g., Primary Sweep).
4. Click **Add Trace**.

A trace is added to the traces list under its report icon in the **Project Manager** window. The trace represents the function of the quantity you selected and is plotted against other quantities or swept variable values. Selecting a Trace in the **Project Manager** window 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 on the **Project Tree**.

By the default, the Trace name is the definition (the category, quantity and function). The trace is 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** window. 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 2D and Circuit Trace Properties

To edit trace properties such as the name, the component definition, or 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** window.

   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** window, 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 a **trace component definition**:

1. Select the trace in the **Project Manager** window.
2. In the docked Properties window for the trace, select the component field of interest and choose **Edit** on the drop-down menu.

   This displays the an edit Component field window.form which Edit the category, quantity and function.

3. Click OK to apply the changes and close the window.

To edit a **trace Context**:

1. Select the trace in the **Project Manager** window to open 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 on the drop-down menu.

To edit a **variable** for a trace:

1. Select the trace in the **Project Manager** window to open the docked properties window.
2. Under the -Variables category, on the Families line, click **Edit** to open the Edit families window.

   From this window, select **Sweeps** or **Variations**. Each selection changes the available editable values.

   If other nominal values are available, click the **[...]** button to select from a list.

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 (2D and Circuit)

To edit the display properties of a trace:

1. Select a trace in an open **Report** window.
2. Clickce on the trace to view a **Docked Properties** window, or double-click to open **Properties** window.

The display properties window for a 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 on the Report. It shows the characteristics of the trace as defined in the **Report** window. To edit a trace name, see Editing Trace Properties
- Color — shows the Trace color. Double-click to open a Color window. You can select from Basic colors, or custom colors. You can define up to 16 custom colors by selecting or by editing the Hue, Saturation, Luminescence, and the Red, Green, and Blue values.
- Line style — a drop-down menu lets you select Solid, Dot, Dash, or Dot-dash.
- Line width — a 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.
- Show Symbol — whether to show a symbol at the data points on the line.
- Symbol Frequency — how often to show symbols on the trace.
- 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 — click the check box to set the symbol display as a solid or as hollow.
- Symbol Arrows — click the check box to use arrows on the curve ends

> **Note:**
>
> So curves with single points always appear, Box is the default symbol. None cannot be selected as 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.

3. Edit the properties of interest and OK the Properties window to apply the changes and close the window.

Adding Data Markers to 2D and Circuit Traces

The Reporter includes **Report 2D > Marker** menu commands and toolbar icons

that 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, 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 open 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 a movable marker at the origin of the plot with a vertical line rising on the X axis. To move an X marker, click the X label and drag it to the appropriate 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. This marker is not cleared by the **Clear All** command, and must be deleted by selecting it and using the **Edit Delete** command.
- **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.

- **Marker** – this command lets you place a marker at an arbitrary point on a selected trace.
- **X Marker** – this command adds a movable marker at the origin of the plot with a vertical line rising on the X axis. To move an X marker, click the X label and drag it to the appropriate 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. This marker is not cleared by the **Clear All** command, and must be deleted by selecting it and using the **Edit Delete** command.
- **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.:

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.
- **Next Data Point** (Right) — moves a selected X marker to the next data point.
- **Previous Data Point** (Left) — moves a selected X marker to the previous 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 except X Markers.

Discarding 2D and Circuit Report Values below a Specified Threshold

To prevent real small numbers from skewing a plot, you can discard small values (under 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

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 the **Axis** tab, drag 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 field is inactive if the Specify Discard Values check box is not enabled.
5. Click **OK** to apply the Discard Values to the report.

Add 2D or Circuit Trace Characteristics

Add or clear additional characteristics to a selected trace. To add additional characteristics to a selected trace:

1. Select a trace in a report plot or legend.
2. Click **Report 2D** > **Trace Characteristics**, or right-click the selected trace to open the shortcut menu.
3. Select **Trace Characteristics** > **Add**

This displays the **Add Trace Characteristics** window.

4. Select the **Category** and an associated Function to apply. The available categories depend on the plot, and Category enables the display of associated functions.

Category	Functions for the Category
Math	max, min, pk2pk, rms, avg, integ, integabs, avgabs, rmsAC, ripple, pkavg, XatYMin, XatYMax, XatYVal, cmplx()
PulseWidth	pulsefall9010, pulsefront9010, pulsefront3090, pulsemax, pulsemaxtime, pulsemin, pulsemintime, pulsetail50, pulsewidth5050, pw_plus, pw_plus_max, pw_plus_min, pw_plus_avg, pw_plus_rms, pw_minus_max, pw_minus_min, pw_minus_avg, 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
Radiation	xdb10bandwidth, xdb20bandwidth, lSidelobeX, lSidelobeY, rSidelobeX, rSidelobeY

Given a selected Function, and Category, the **Add Trace** window displays a field that explains the Purpose of the function. For a full list of functions and their definitions, see Selecting a Function.

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. Save these values by clicking **Default**.
6. Click **Add** to add the specified characteristics to the Trace.

To remove existing trace characteristics:

1. Select a trace in a report plot or legend.
2. Click **Report 2D > Trace Characteristics**, or right-click the selected trace to open the shortcut menu.
3. Select **Trace Characteristics > Clear All**

Trace characteristics are clear on the selected trace.

Removing Traces from 2D and Circuit Reports

You can remove traces on the traces list in the following ways:

To *remove one trace* on the report:

- Select the trace you want to remove on the Project tree, then click **Delete**.

To *remove all traces* on the report:

- Select all the traces and click **Delete**.

Define Traces Using Range Functions (2D and Circuit)

Range Functions are special functions that use a 2D dataset as input, along with 0 or more additional parameters. Range functions can be used to produce a user-created 2D report display that is a collection of traces and their attributes. The trace collection and attributes are used to generate a portion of a report-definition that generates a family of curves in the report window.

Range Functions can be applied to a range (subset) of points on an X-Y plot, then calculates a single-number representation of the specified range, and display that number directly over a plotted wave-form. Range functions can be used to extract trace characteristics (such as max, min, overshoot) from a plot and use those values for additional plotting, or use the values for exporting to a file, where the data table formats are supported. Range Functions are listed in the **Reporter** window and **Output Variable** window for users to employ. Extracted trace characteristics may also be used in optimetrics.

For example, with a wave-form in a transient plot that contains a square pulse, a range function can calculate and display a single value that represents the High-to-Low/Low-to-High transition states. Or, with a plot that shows a pulse, the PulseWidth and RiseTime Range Functions can use the entire plotted curve to calculate a single number that represents the width or rise-time, and display that single-number representation directly over the specified range of plotted points.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Range functions trace characteristics are displayed on the report window as a column in legend window
- Range functions are available from reporter and optimetrics
- Numerous range functions are available in the following categories:

— Pulse width functions

— Overshoot/Undershoot functions

— TR & DC functions

— Error functions

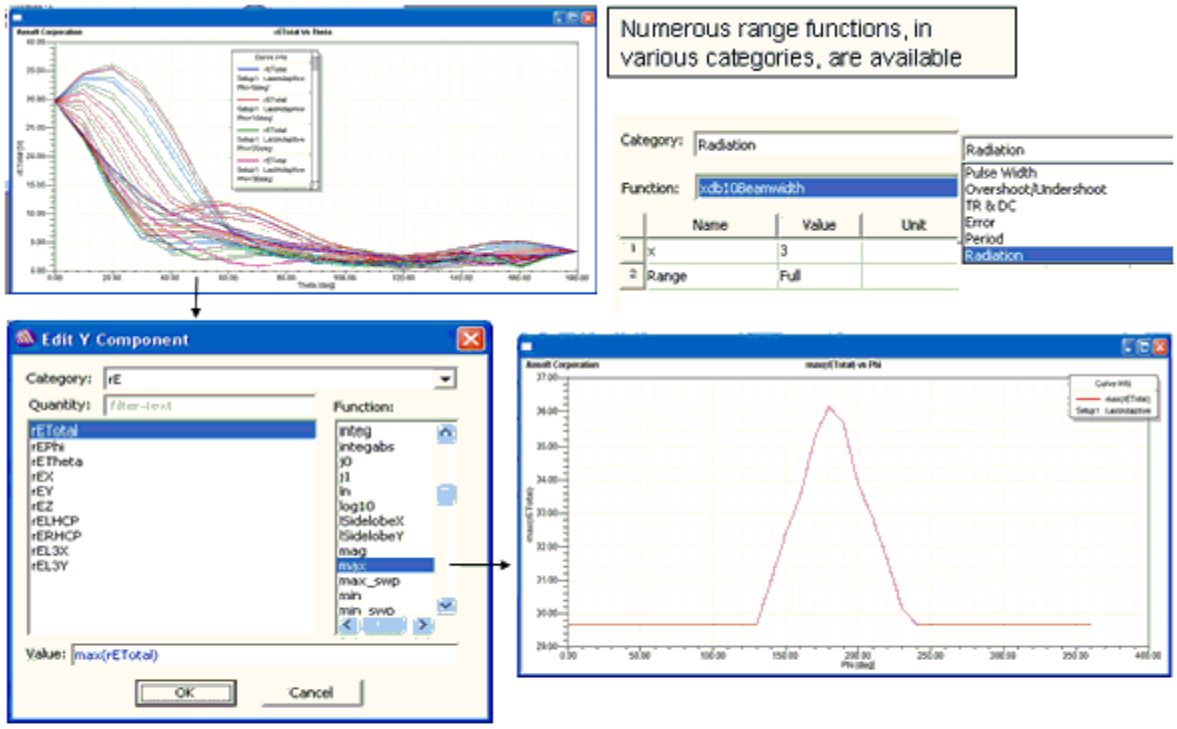
— Period functions

— Radiation functions

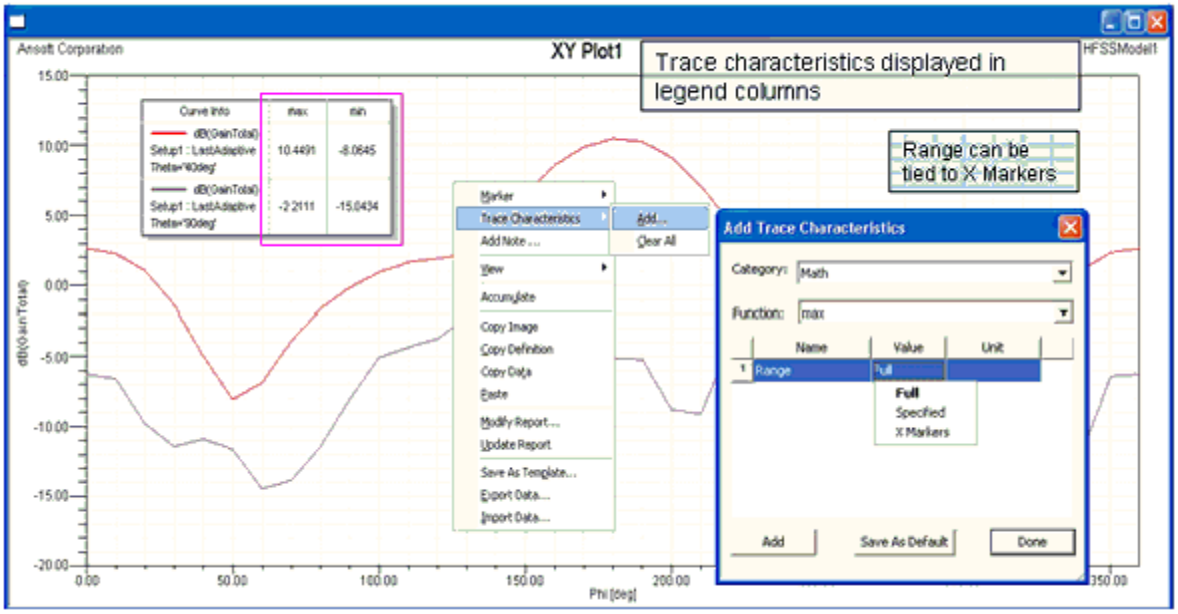
> **Note:**
>
> For all range functions, the second parameter is assumed to be a sweep variable, and if it is not, the following error message results: "**Second parameter to functions that apply to a range of values must be a sweep variable** ". You receives this message if you create a range function where the second parameter is a non-variable, such as deriv(dB(S11),dB(S12)). Whereas deriv(dB(S11),F) is acceptable.

Define Traces Using Range Functions

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Trace Characteristics



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 the available Range Functions:

Range Function	Category	Description	Parameter(s)
avg	Math	Average of first param over the second param.	N/A
avgabs	Math	Returns the mean of the absolute value of the selected quantity.	N/A
bandwidth	AC	Returns the bandwidth of the selected simulation quantity in Hz.	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
cang_deg	TR & DC	Cumulative angle of the first param in degrees, along the second param (typically sweep variable). Returns double.	N/A
cang_rad	TR & DC	Cumulative angle of the first param in radians, along the second param (typically sweep variable). Returns double.	N/A
crestfactor	TR & DC	Returns the crest factor (peak/RMS) for the selected quantity.	N/A
deadtime	TR & DC	Obtains the latest time when the qtyl is within a tolerance of zero.	Tolerance:The +/- bandwidth around 0
delaytime	TR & DC	Obtains the time from zero to 50% of the target point.	Target: The target value for input
deriv	TR & DC	Derivative of a given param	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.

distortion	TR & DC	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.	Frequency: Freq in Hz at which to calculate the RMS of the qty
duty	TR & DC	Duty cycle measurement.	N/A
eyeheight	TR & DC	Eye-diagram characteristics	N/A
eyejitter	TR & DC	Eye-diagram characteristics	N/A
eyewidth	TR & DC	Eye-diagram characteristics	N/A
FFT	Math	Fast Fourier Transform; returns output signal.	Incoming signal.
formfactor	TR & DC	Returns the form factor (RMS/Mean Absolute Value) for the selected quantity.	N/A
fundamentalmag	TR & DC	Returns the RMS value of the fundamental frequency for the selected quantity and an additional argument frequency, which specifies the fundamental frequency.	Frequency: Freq in Hz at which to calculate the RMS of the qty
gaincrossover	AC	Returns the gain crossover	Base Gain: Ref AC magnitude; Base

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		frequency (where the gain is 0 dB) of the selected simulation quantity in Hz.	phase: Ref phase angle
gainmargin	AC	Returns the gain margin in dB at the phase crossover frequency of the selected simulation quantity.	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
iae	Error	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument.	Target: Target value
integ	Math	Integral of the selected quantity. Uses trapezoidal area.	N/A
integabs	Math	Returns the integral of the absolute value of the selected qty.	N/A
ise	Error	Returns the integral of the squared deviation of the selected quantity from a target value that is entered via an additional argument.	Target: Target value
itae	Error	Returns the time-weighted absolute deviation of the selected quantity from a target value that is entered via an additional argument.	Target: Target value

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

itse	Error	Returns the time-weighted squared deviation of the selected qty from a target value that is entered via an additional argument.	Target: Target value
lowercutoff	AC	Returns the lower 3dB frequency of the selected simulation channel in Hz.	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
max	Math	Maximum (of magnitudes).	N/A
min	Math	Minimum (of magnitudes)	N/A
overshoot	Overshoot/Undershoot (Overridable)	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range.	Threshold: The reference value from where the overshoot/undershoot is calculated; Number of Points: Number of evenly spaced time points
peakgain	AC	Returns the peak gain of the selected simulation quantity in dB.	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
peakgainfreq	AC	Returns the frequency in Hz at which the peak gain of the selected simulation quantity occurs.	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
per	Period (Overridable)	Calculates period.	Threshold:Y transition value which determines the period; Number of Points: Number of evenly

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			spaced time points
phasecrossover	AC	Returns the phase crossover freq, at which the phase is -180 degrees, in Hz for the selected simulation quantity.	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
phasemargin	AC	Returns the phase angle in degrees at the gain crossover frequency of the selected simulation quantity.	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
pk2pk	Math	Difference between max and min of the first param over the second param. Returns the peak-to-peak value for the selected simulation quantity.	N/A
pkavg	Math	Returns the ratio between the peak-to-peak value and the average value of the selected quantity	N/A
pmax	Period (Overridable)	Maximum period of input stream.	Threshold:Y transition value which determines the period; Number of Points: Number of evenly spaced time points
pmin	Period (Overridable)	Minimum period of input stream.	Threshold:Y transition value which determines the period; Number of Points: Number of evenly spaced time points
prms	Period (Overridable)	Rms of period of input stream.	Threshold:Y transition value which

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			determines the period; Number of Points: Number of evenly spaced time points
pulsefall9010	Pulse Width	Returns the pulse fall time of the selected quantity derived from estimates of the signal at 90% and 10% of its peak value.	N/A
pulsefront1090	Pulse Width	Returns the pulse front time of the selected quantity derived from estimates of the signal at 10% and 90% of its peak value.	N/A
pulsefront3090	Pulse Width	Returns the pulse front time of the selected quantity derived from estimates of the signal at 30% and 90% of its peak value.	N/A
pulsemax	Pulse Width	Returns the pulse maximum on the front and tail estimates for the selected quantity.	N/A
pulsemaxtime	Pulse Width	Returns the time at which the maximum pulse value of the selected quantity is reached.	N/A
pulsemin	Pulse Width	Returns the pulse minimum on the front and tail estimates for the	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.

		selected quantity.	
pulsemintime	Pulse Width	Returns the time at which the minimum pulse value of the selected quantity is reached.	N/A
pulsetail50	Pulse Width	Returns the pulse tail time of the selected quantity on the virtual peak to 50%.	N/A
pulsewidth5050	Pulse Width	Returns the pulse width of the selected quantity as measured on the 50% points on the pulse front and pulse tail.	N/A
pw_minus	Pulse Width (Overridable)	Pulse width of the first negative pulse.	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_minus_avg	Pulse Width (Overridable)	Average of the negative pulse width input stream.	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_minus_max	Pulse Width (Overridable)	Max. Pulse width of the negative pulse of input stream.	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_minus_min	Pulse Width (Overridable)	Min. Pulse width of the negative pulse of input stream.	Threshold:Y transition value which determines the pulse width ; 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.

			Points: Number of evenly spaced time points
pw_minus_rms	Pulse Width (Overridable)	RMS of the negative pulse width input stream	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_plus	Pulse Width (Overridable)	Pulse width of first positive pulse.	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_plus_avg	Pulse Width (Overridable)	Average of the positive pulse width input stream	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_plus_max	Pulse Width (Overridable)	Max. Pulse width of the positive pulse of input stream	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_plus_min	Pulse Width (Overridable)	Min. Pulse width of the positive pulse of input stream	Threshold:Y transition value which determines the pulse width ; Number of Points: Number of evenly spaced time points
pw_plus_rms	Pulse Width (Overridable)	RMS of the positive pulse width input stream	Threshold:Y transition value which determines the pulse width ; Number of Points: 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.

			evenly spaced time points
ripple	Math	Returns the ripple factor (AC RMS/Mean) for the selected quantity.	N/A
risetime	TR & DC	Obtains the time taken to go from 10% to 90% of target point	Target: The target value for input
rms	Math	Returns total RMS of the selected quantity	N/A
rmsAC	Math	Returns the AC RMS for the selected quantity	N/A
settlingtime	TR & DC	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.	Target; Tolerance
undershoot	Overshoot/Undershoot (Overridable)	Calculates peak undershoot given a threshold value and number of evenly spaced points over entire time range	Threshold: The reference value from where the overshoot/undershoot is calculated; Number of Points: Number of evenly spaced time points
uppercutoff	AC	Returns the upper 3dB frequency of the selected simulation channel in Hz	Base Gain: Ref AC magnitude; Base phase: Ref phase angle
XAtYMax	Math	Returns the X value at maximum Y	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.

XAtYMin	Math	Returns the X value at minimum Y	N/A
XAtYVal	Math	Returns the X value at the first occurrence of Y value	Y Value: Y value at which to find X

Copy and Paste of 2D and Circuit 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 is evaluated in the context of the target design or report.

> **Note:** If the report or trace definition contains properties that do not exist in the target design (e.g., a port name) an error is posted that indicates a solution does not exist for this trace.

> **Note:** 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:

From the **Project Manager** window, expand the **Results** folder. Then right-click the report and select **Copy Definition**.

To Paste a Report Definition:

From the **Project Manager** window, right-click **Results** and select **Paste**. A new report is created and it contains the copied definitions.

To Copy an individual Trace Definition(s):

From the **Project Manager** window, expand the **Results** folder and the report. Then right-click the trace or traces and select **Copy Definition**.

To paste the Trace Definition(s):

Right-click the report in the target design to copy the trace or traces and select **Paste**.

A new trace(s) is added to the report and it contains the copied trace definition(s).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you copy and paste a report or trace definition to a design which contains a definition with the same name, then an incremented number is appended to the pasted report or trace name.

Copy and Paste 2D and Circuit 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 in the report or trace are copied and pasted to the target design or report.

To copy all data from a report:

Right-click the report name in the **Project Manager** window and select **Copy Data**, or use the menu bar **Edit > Copy Data**, or right-click within a plot to display and select **Copy Data**.

To paste copied report data:

Right-click **Results** in the **Project Manager** window 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 Manager** window and select **Copy Data**.

To paste copied trace data:

Right-click the report in the target design to copy the trace data to 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 is appended to the pasted name.

Variables, Quantities, and Functions in 2D and Circuit Reports

The follow topics describes how to sweep variables, and select various quantities and functions for use in specifying values to be displayed in plots and reports.

Sweeping a Variable in a 2D or Circuit Report

In Ansys Electronics Desktop, 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To specify the swept variable values to plot a selected quantity against:

1. In the **Report** window, select the variable on the X (Primary Sweep) drop-down menu.
2. To modify the values that is plotted for a variable:
   a. Click **[...]** on the **X (Primary Sweep)** line of the **Report** window to display a list of the possible values.
   b. Select **All Values** or click **Edited** to display a window that lets you specify the sweeps to use.

   All of the selected variable's values is plotted.

2D and Circuit Selecting a Function

The value of a quantity being plotted depends upon its mathematical function, which you select on the**Trace** tab **Function** group box in the **Report**window. 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").

The following functions listed can be applied as **Range Functions** to previously specified Quantities and Functions using the **Set Range Function** window. Some of these functions can operate along an entire curve (min, max, integ, avg, rms, pk2pk, cang_deg, cang_rad). Their syntax is as follows:

- cang_deg(*quantity*) implicitly implies derivative over the primary sweep
- cang_deg(*quantity*, *SweepVariable*) explicitly means derivative over the sweep variable specified in the second argument (such as "Freq")

> **Note:** For all range functions, the second parameter is assumed to be a sweep variable, and if it is not, the following error message results: "**Second parameter to functions that apply to a range of values must be a sweep variable** ". You receive this message if you create a range function where the second parameter is a non-variable, such as deriv(dB(S_{11}),dB(S_{12})). Whereas deriv(dB(S_{11}),F) is acceptable.

You can select on the following functions in the **Trace** tab **Function** group box:

abs	Absolute value
acos	Arc cosine
acosh	Hyperbolic arc cosine
ang_deg	Angle (phase) of a complex number, cut at +/-180
ang_deg_val	Angle (phase of a complex number in unitless degree values. Returns

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	simple numbers.
ang_rad	Angle in radians
asin	Arc sine
asinh	Hyperbolic arc sine
atan	Arc tangent
atanh	Hyperbolic arc tangent
avg	Average of first parameter over the second parameter
avgabs	Absolute value of average.
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.
cang_deg_val	Cumulative angle (phase) of the first parameter of the selected simulation quantity in unitless degree values. Returns simple numbers.
cang_rad	Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable) Returns a double precision value.
cmplx	Complex number with the first parameter as the real part and the second parameter as the imaginary part.
conjg	Conjugate of the complex number.
cos	Cosine
cosh	Hyperbolic cosine
crestfactor	Peak/RMS (root mean square) for the selected simulation quantity
dB(x)	20*log10(|x|)
dBm(x)	10*log10(|x|) +30
dBW(x)	10*log10(|x|)
db10normalize	10*log [normalize(mag(x))]
db20normalize	20*log [normalize(mag(x))]
deriv	Derivative of a given parameter.
even	Returns 1 if integer part of the number is even; returns 0 otherwise
exp	Exponential function (the natural anti-logarithm)
formfactor	Returns root mean square RMS/Mean Absolute Value for the selected simulation quantity.
iae	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument. To use this function, you need to open the Add Trace Characteristics window and select the Error category.
im	Imaginary part of the complex number
int	Truncated integer function

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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. To use this function, you need to open the Add Trace Characteristics window and select the Error category.
itae	Returns the time-weighted absolute 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 window and select the Error category.
itse	Returns the time-weighted squared deviation of the selected qty from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics window 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)
ln	Natural logarithm
log10	Logarithm base 10
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	Maximum of magnitudes.
max_swp	Maximum value of a sweep.
min	Minimum magnitudes.
min_swp	Minimum value of a sweep.
nint	Nearest integer
normalize	Divides each value within a trace by the maximum value of the trace. ex. normalize(mag(x))
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise
overshoot	Obtains the peak overshoot over a point (double argument)
per	Calculates period.
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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	value of the selected quantity
pmax	Period max
pmin	Period minimum
prms	Period Root Mean Square.
pulsefall9010	Pulse fall time of the selected simulation quantity according to the 90%-10% estimate.
pulsefront9010	Pulse front time of the selected simulation quantity according to the 10%-90% estimate.
pulsefront3090	Pulse front time of the selected simulation quantity according to the 30%-90% estimate.
pulsemax	Pulse maximum on the front and tail estimates for the selected simulation quantity.
pulsemaxtime	Time at which the maximum pulse value of the selected simulation quantity is reached.
pulsemin	Pulse minimum on the front and tail estimates for the selected simulation quantity.
pulsemintime	Time at which the minimum pulse value of the selected simulation quantity is reached.
pulsetail50	Pulse tail time of the selected simulation quantity on the virtual peak to 50%.
pulsewidth5050	Pulse width of the selected simulation quantity as measured on the 50% points on the pulse front and pulse tail.
PulseWidth Functions	
pw_plus	Pulse width of first positive pulse
pw_plus_max	Max. Pulse width of input stream
pw_plus_min	Min. Pulse width of input stream
pw_plus_avg	Average of the positive pulse width input stream
pw_plus_rms	RMS of the positive pulse width input stream
pw_minus_max	Max. Pulse width of input stream
pw_minus_min	Min. Pulse width of input stream
pw_minus_avg	Average of the negative pulse width input stream
pw_minus_rms	RMS of the negative pulse width input stream
polar	Converts the complex number in rectangular to polar
re	Real part of the complex number
rect	Converts the complex number in polar to rectangular
rem	Fractional part
ripple	Returns the ripple factor (AC RMS/Mean) for 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.

rms	Returns total root mean square of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
rsidelobex	The ‘x’ value for the right side lobe: the next highest value to the right of the max value.
rsidelobey	The ‘y’ value for the right side lobe: the next highest value to the right of the max value.
sgn	Sign extraction
sin	Sine
sinh	Hyperbolic sine
sqrt	Square root
tan	Tangent
tanh	Hyperbolic tangent
Undershoot	Obtains the peak undershoot over a point (double argument).
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
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 window 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 window and select the Radiation category.
y0	Bessel function of the second kind (0^{th} order)
y1	Bessel function of the second kind (1^{st} order)

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, S_{11} say, 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(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 so by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val(S_{11}) is 90 and "1 + ang_deg_val(S_{11})" is 91.

Selecting Solution Quantities to Plot (2D and Circuit)

When you create a report of Modal or Terminal solution data, 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 **Electronics Desktop** such as S_{11}, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta. The valid categories available depend on the type of quantity (real or complex) that is being plotted, the setup, the solution type, and the plot domain.

To select an S-parameter quantity 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. In the **Report** window, **Trace** tab, select one of the following categories:

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 on the original field solution.
S-parameter	S-parameters on the S-matrix. For designs which include a Frequency Selective Surface (FSS)-referenced radiation boundary, S11 and S21 represent the extracted reflection and transmission coefficients, respectively.
Y-parameter	Admittance matrix parameters computed on the S-parameters and port impedances.
Z-parameter	Impedance matrix parameters computed on the S-parameters and port impedances.
Gamma	Propagation constants for the S-parameters.
Port Z_o	Characteristic port impedances.
VSWR	Voltage standing wave ratio, calculated on the equation $\frac{1+\lvert S_{ij}\rvert}{1-\lvert S_{ij}\rvert}$.

2. Select a quantity to plot on the **Quantity** group box. The available quantities depends upon the selected category and the setup of the design.

Selecting a Field Quantity to Plot (2D and Circuit)

When plotting field quantities, the quantity can be a value that was calculated by **Electronics Desktop** 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."
2. In the **Report** window, select Geometry for the Context, unless you are plotting scalar (e.g., integration).
3. In the **Report** window, select one of the following categories:

Variables	Intrinsic variables, such as frequency or phase, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	user-defined expressions applied to derive quantities on the original field solution.
Calculator	Includes scalar and vector field quantities automatically calculated by

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expressions Ansys Electronics Desktop, 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 on the **Quantity** group box. The available quantities depends upon the selected category and the setup of the design.

2D and Circuit Selecting a Far-Field Quantity to Plot

When plotting far-field quantities, the quantity can be a value that was calculated by **Electronics Desktop** such as antenna gain, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.

To select a far-field quantity to plot:

1. When you create the report, specify the Report Type as "Far Fields."
2. In the **Report** window, select one of the following categories for the field setup:

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 on the original field solution.
Gain	Gain is 4π times the ratio of an antenna's radiation intensity in a given direction to the total power accepted by the antenna.
Axial Ratio	Axial ratio of the electric field.
Antenna Params	Ansys Electronics Desktop-calculated quantities that include peak directivity, radiated power, accepted power, radiation efficiency, max U, and array factors. For far-field setups, the decay factor for lossy materials is calculated as a constant for all far fields.
Radar Cross-Section (Bistatic RCS)	The radar cross-section (RCS) or echo area (σ) is measured in meters squared and represented for a bistatic arrangement (i.e., when the transmitter and receiver are in different locations). This is represented by: $\sigma = \frac{4\pi r^2 \lvert E_{\text{scat}} \rvert^2}{\lvert E_{\text{inc}} \rvert^2}$ where: • E_{scat} is the scattered E-field. • E_{inc} is the incident E-field. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.*

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D and Circuit Directive Gain

Directive Gain is the ratio of the radiation intensity at a specified observation angle to the average radiation intensity. For the observation angle (q,f), the directivity is:

$$D(\theta,\phi) = \frac{U(\theta,\phi)}{U_{ave}} = \frac{U(\theta,\phi)}{\frac{1}{4\pi}\oint\oint U(\theta,\phi)d\Omega}$$

where U is the radiation intensity, which is defined as:

$$U(\theta,\phi) = \frac{1}{2}\mathrm{Rc}(\vec{E}\times\vec{H})\cdot r^2\hat{r}$$

The average radiation intensity is related to the radiated power by:

$$U_{ave} = \frac{P_r}{4\pi}$$

Directivity

Directivity is the maximum value of Directive Gain, or:

$$D = \frac{U_{max}}{U_{ave}}$$

Reference

Warren Stutzman and Gary Thiele, *Antenna Theory and Design*, Wiley, 1981.

2D and Circuit Radiation Efficiency

The radiation efficiency is the ratio of the radiated power to the accepted power given by:

$$e = \frac{P_{\mathrm{rad}}}{P_{\mathrm{acc}}}$$

where:

- P_{rad} is the radiated power in watts
- P_{acc} is the accepted power in watts

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 the radiation efficiency is calculated on the accepted power, a port must be defined for radiation efficiency to be displayed.

Setting a Range Function (2D and Circuit)

To apply a range function to the Y, Z, or Mag component of a trace:

1. Click **Range Function** in the **Reports** window.

   This opens the **Set Range Function** window. The functions available are the same as described in Selecting a Function.

2. Click the Specified radio button on the Range function line.

   This enables the Range Function fields.

3. Select the **Category**, then an associated Function to apply. The available categories depend on the plot, and Category enables the display of associated functions.

Category	Functions for the Category
Math	max, min, pk2pk, rms, avg, integ, integabs, avgabs, rmsAC, ripple, pkavg, XatYMin, XatYMax, XatYVal
PulseWidth	pulsefall9010, pulsefront9010, pulsefront3090, pulsemax, pulsemaxtime, pulsemin, pulsemintime, pulsetail50, pulsewidth5050, pw_plus, pw_plus_max, pw_plus_min, pw_plus_avg, pw_plus_rms, pw_minus_max, pw_minus_min, pw_minus_avg, 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
Radiation	xdb10bandwidth, xdb20bandwidth, lSidelobeX, lSidelobeY, rSidelobeX, rSidelobeY
Eye Measurements	EyeLevelZero, EyeLevelOne, EyeAmplitude, EyeHeight, EyeSignalToNoise, EyeOpeningFactor, EyeWidth, EyeJitterP2P, EyeJitterRMS, EyeRiseTime, EyeFallTime

Given a selected Function, and Category, the **Set Range Function** window displays a field that explains the Purpose of the function. For a full list of functions and their definitions, see Selecting a Function.

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 causes the display of a description in the **Purpose** field. If the function requires a value (such as the XatYVal Math function or the pw_minus_max Pulse Width function), the following table the function field displays the name, editable value field, unit, and description.

The Eye Measurement functions have five editable parameters.

4. Click the **Over Sweep** drop-down menu to select from available sweeps.
5. To select from available Sweeps, or to edit them, click **[...]** and uncheck **Use All Sweeps**.

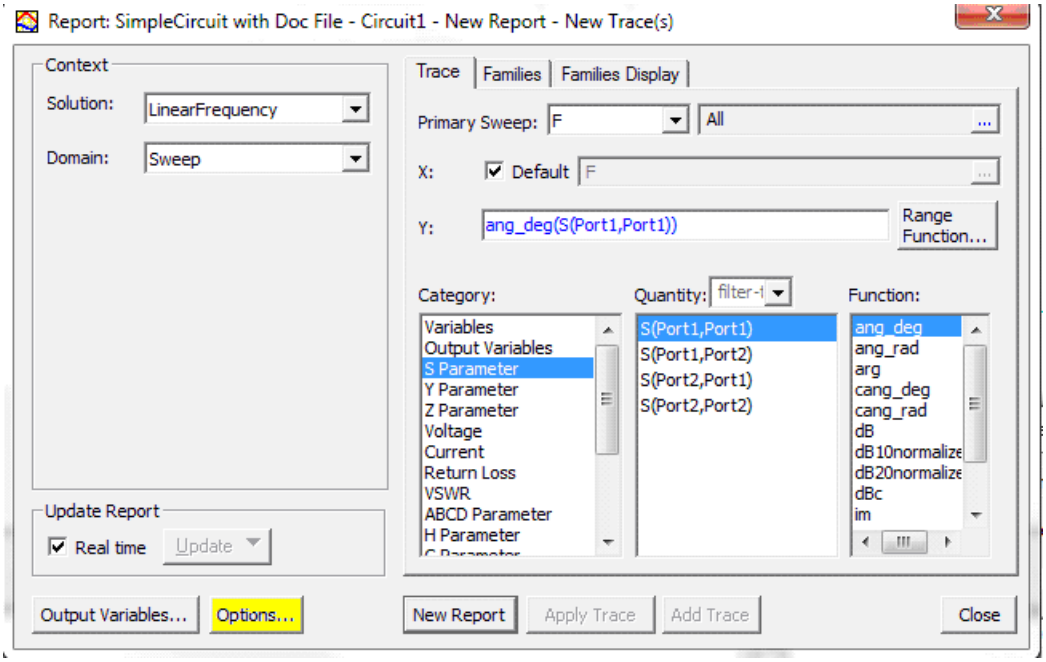
This enables a list of the sweeps. The sweep(s) you select is displayed on the Over Sweep line. You can use the buttons to **Clear All Selections** or **Select All** sweeps.

6. Select the Sweeps **Default** or **Edited** buttons to specify whether to accept the default or edited sweeps.
7. To edit the sweeps further, select **[...]** to open the **Edit Sweep** window.

For frequency variables, this lets you specify a single value, linear step, linear count, decade count, or octave 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.

Handling a Large Number of 2D or Circuit Ports



The tree view is only displayed when the following environment variables are configured.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ANSYS_ENABLE_LARGE_PORTS = 1

  Required for Matrix parameters to display in tree view when the number of ports > 10.
- ANSYS_DISABLE_TREE_LIMIT = 1

  Required for Matrix parameters to display in tree view when the number of ports <= 10.

Working with Eye Diagrams

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. For more information about setting up eye diagram report types, see the following topics:

- Eye Diagram Report
- Stacked Eye Diagram Report
- Statistical Eye Diagram Report

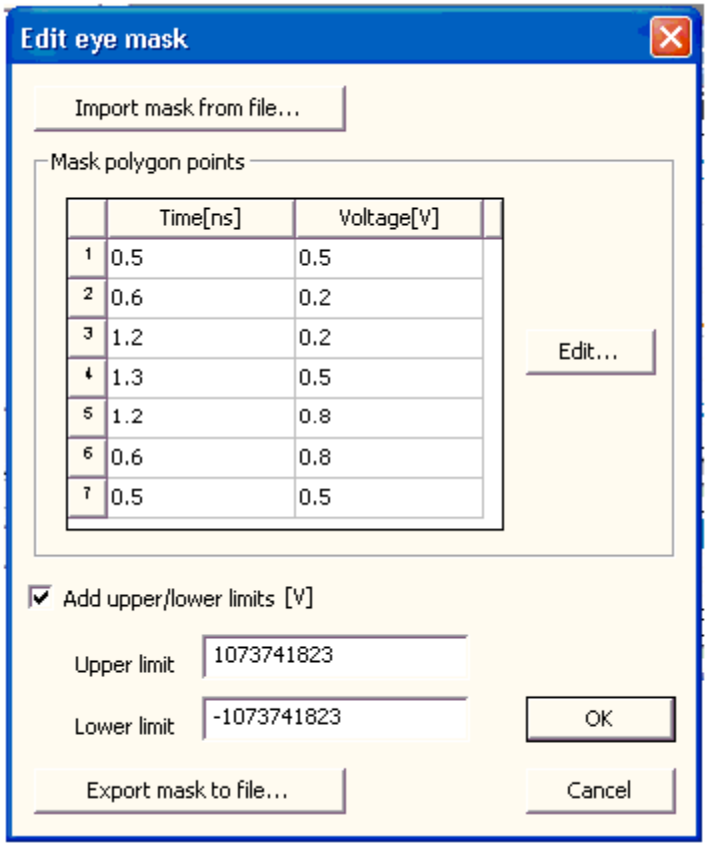
Modifying Circuit Eye Diagram Reports

Double-click the Report to open a Property window with several tabs.

- Click the **General** tab to set the background color, plot area, and other general parameters.
- Click the **Mask** tab to enable and define a mask or keep-out region that is overlaid on the eye diagram. Click **Show Mask** to enable the edit fields. You can specify the fill color, boundary color, and transparency.

Click **Edit** on the **Mask** tab to open the Edit Eye Mask window:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

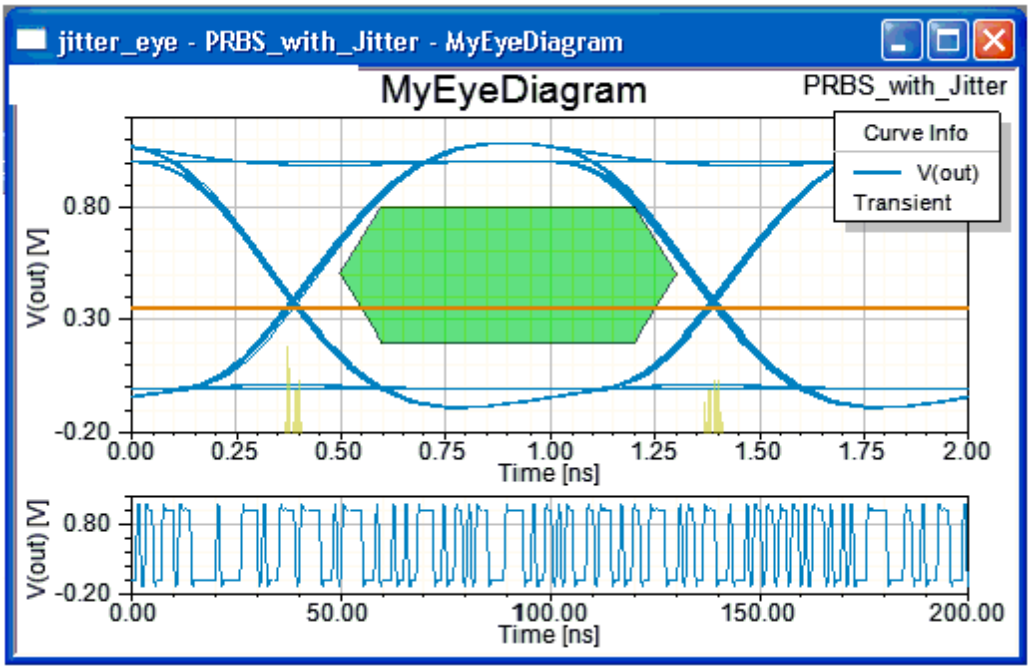


Use the window to specify edit points that define the mask polygon and upper and lower limits as applicable. If appropriate, save the mask setting to a file.

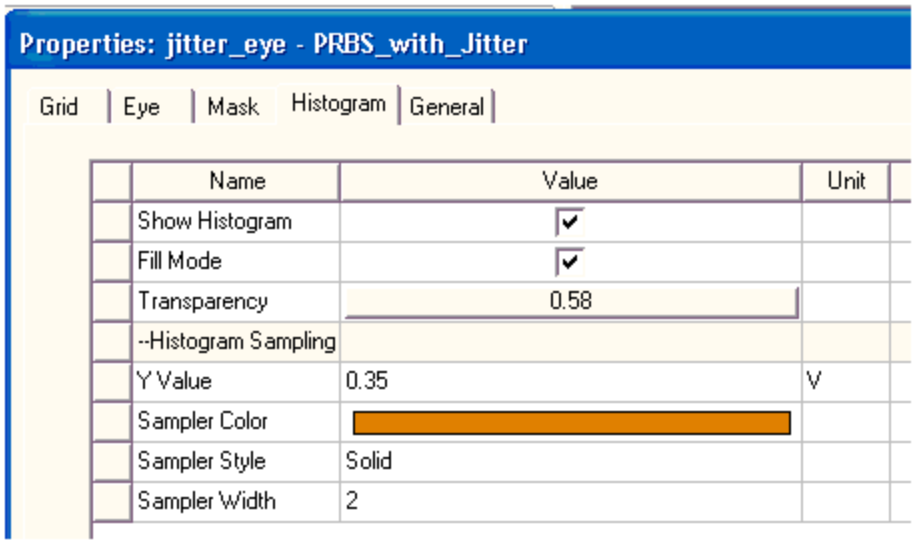
Click **OK** to open the mask on the eye diagram.

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 how the mask appears on the graph:



- Click the **Histogram** tab to control the display of a histogram that counts the number of crossings of a specified Y-value. In the illustration above, the histogram is in yellow and the Y-value is the orange line. Here is the **Properties** page showing the histogram setup fields:

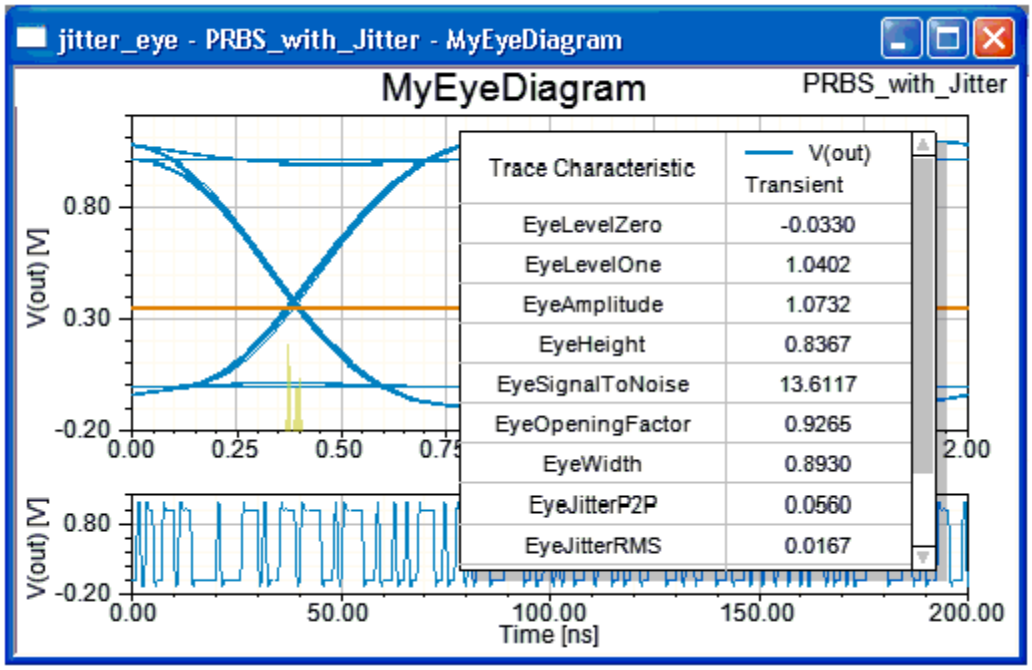


Adding Trace Data to the Eye Diagram Report

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 and select **Trace Characteristics > Add** to open the **Trace Characteristics** window. Select a **Category** of data, then select a **Function** within that category. Click **Add**. The value of the trace characteristic is added to the report display.

Right-click the Report and select **Trace Characteristics > AddAll Eye Measurements** on the menu to open the trace measurements in tabular format. Here is the result:



Eye Diagram References

A tutorial on interpreting eye diagrams may be found at:

`http://www.complextoreal.com/chapters/eye.pdf`

Other references on eye diagrams may be found at:

`http://www.scientificarts.com/logo/logos.html`

`http://www.findarticles.com/p/articles/mi_m0HPJ/is_n4_v45/ai_16226058`

Eye Measurements (2D and Circuit)

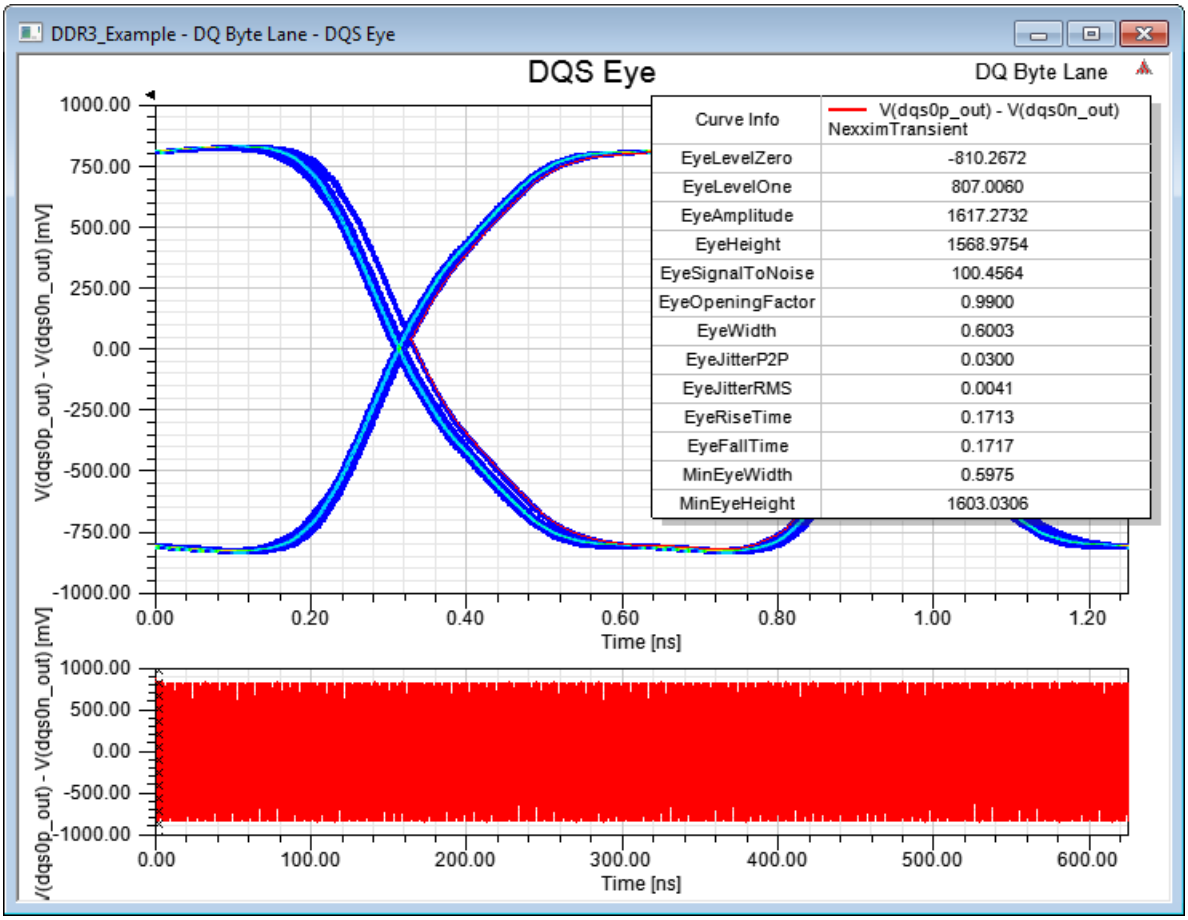
Several standard eye-diagram measurements are provided in **Electronics Desktop**. The following eye measurements are supported:

- EyeLevelZero
- EyeLevelOne
- EyeAmplitude

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- EyeHeight
- EyeSignalToNoise
- EyeOpeningFactor
- EyeWidth
- EyeJitterP2P
- EyeJitterRMS
- EyeRiseTime
- EyeFallTime
- MinEyeWidth
- MinEyeHeight

You can right-click the report and select **Trace Characteristics > AddAll Eye Measurements** to display all the eye measurements in tabular format.



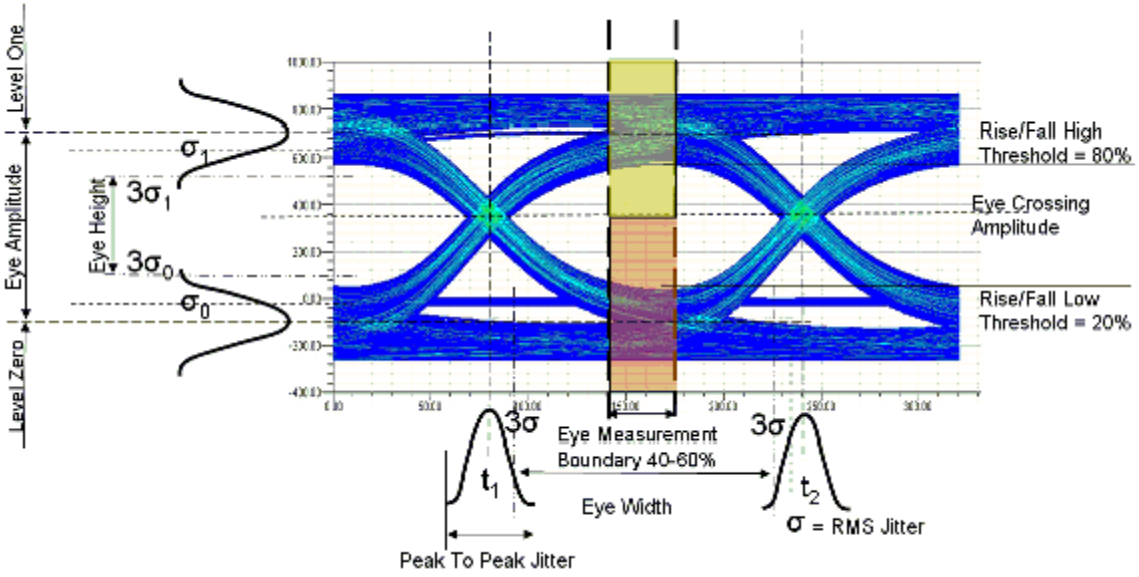
To add an eye measurement, right-click the eye diagram and select one of the following:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Trace Characteristics/Add** to add eye measurements individually
- Select **Trace Characteristics/Add All Eye Measurements** to add all available eye measurements

To remove an eye measurement, right-click the eye diagram:

- Select **Trace Characteristics/Clear All**



Values in the Eye Measurements Table are Calculated as Follows:

- **EyeLevelZero** and **EyeLevelOne** are the mean values of the lower/upper vertical histograms in the 40-60% center region of the eye, where “center” is defined by the eye-crossing points t1 and t2. Vertical histograms are created above or below the eye-crossing point (represented as yellow/orange shaded regions in the preceding figure). The mean value of these histograms yields EyeLevelZero and EyeLevelOne.
- **EyeAmplitude** = (EyeAmplitude = EyeLevelOne - EyeLevelZero). This is the voltage value for the eye crossing point, which is computed by creating a vertical histogram at the eye crossing points. Eye Crossing Amplitude is computed as the mean of the histogram. A front group box eye diagram has two eye crossings. Eye Crossing Amplitude is calculated as being the mean of these crossing amplitudes, and is then used to compute other horizontal eye measurements, such as eye width, RMS jitter, and peak-to-peak jitter.
- **EyeHeight** = [(EyeLevelOne - $3\sigma_1$) - (EyeLevelZero + $3\sigma_0$)] where σ_1 and σ_0 are the standard deviations of the vertical histograms used to determine EyeLevelZero and EyeLevelOne.
- **EyeSignalToNoise** = [EyeLevelOne - EyeLevelZero] / (σ_1 + σ_0)
- **EyeOpeningFactor** = [EyeAmplitude - σ_1 - σ_0] / EyeAmplitude

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **EyeWidth** = [(t_2 - 3σ) - (t_1 + 3σ)] where σ is the standard deviation of the horizontal histograms used to determine eye-crossing points.
- **EyeJitterP2P** is the width of the horizontal histogram across the eye-crossing point.
- **EyeJitterRMS** is the standard deviation of the horizontal histogram across the eye-crossing point.
- **EyeRiseTime** and **EyeFallTime** are the differences in mean values of the horizontal histograms centered on the 20% and 80% threshold values. Horizontal histograms are created at the 20% and 80% amplitude thresholds for both rising and falling edges. The difference in the mean values of the histograms for rising edges yields EyeRiseTime, while the difference in mean values for falling edges yields EyeFallTime.
- **MinEyeWidth** is the minimum horizontal opening at the eye-crossing amplitude, typically at the center of the eye. Unlike statistical eye width, if different traces exist for different unit intervals, then MinEyeWidth represents the minimum width of all the traces.
- **MinEyeHeight** is the minimum vertical opening at the eye-measurement point, typically at the center of the eye. Unlike statistical eye height, if different traces exist for different unit intervals, then MinEyeHeight represents the minimum height of all the traces.

Additional Values Not Listed in the Eye Measurements Table:

- **AutoDelay** is the amount of time required to shift the eye diagram so it is centered. You can control this delay either by specifying Manual Delay or allow the system to automatically compute the delay via Auto Delay.
- **AutoComputeCrossAmplitude**

  With Auto Mode “on”, the eye width is calculated as the average of the two peaks of the vertical histogram. This is shown as the "Eye Crossing Amplitude" in the preceding figure. Both statistical and minimum eye widths is calculated at this amplitude.

  With Auto Mode "off", the user can manually set the crossing point voltage to raise/lower where the eye width calculations are performed. Both statistical and minimum eye widths is calculated at this amplitude. The user should enter the voltage value where he thinks the actual crossing point lies--that is, where eye width should be computed (both statistical and MinEyeWidth).
- **AutoComputeEyeMeasurementPoint**

  With Auto Mode “on”, the eye height is calculated as the average of the two eye-crossing points, t1 and t2, as shown in the preceding figure. This point is the Eye Measurement Point. Both statistical and minimum eye heights is calculated at this time point.

  With Auto Mode "off", the user can manually set the crossing point time in order to shift where the eye height calculations are performed. Both statistical and minimum eye heights is calculated at that time point.

  Auto Delay does not affect eye height if the Eye Measurement Point is selected according

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

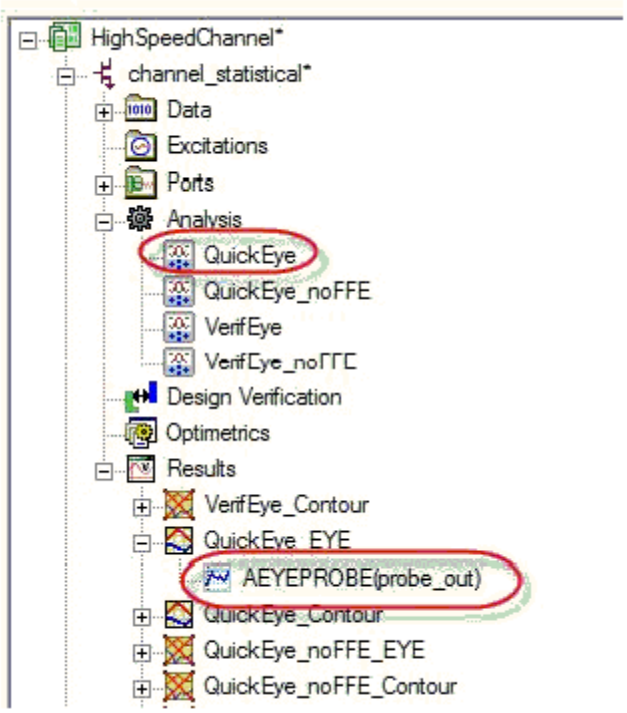
to the eye diagram that is currently displayed. For example, if the UI is 1ns, the front group box eye is drawn over 2ns. With Auto Delay “off”, assuming that the first crossing is at 0.3ns and the second is at 1.3ns, the user should enter 0.8ns as the eye measurement point. With Auto Delay “on”, the first crossing is shown at 0.5ns and second is shown at 1.5ns. In this case, the corresponding eye measurement approximation point is 1ns.

- **EyeCrossingTime** is calculated by creating a horizontal histogram across a small, narrow strip at the middle of the eye (eye-crossing points, t1 and t2). Eye Crossing Time is the mean of the histogram. The middle of the eye is computed as the middle of the vertical extremities of the eye--that is, the midpoint of max & min voltage values across the complete eye diagram. Later, an internal algorithm refines the crossing times by re-computing the crossing points based on a more accurate amplitude. The refined crossing time calculation is then used as the new middle of the eye. A front group box eye diagram has two eye crossings and two Eye Crossing Times.
- **EyeMeasurementPoint** (time) is calculated as the mean of the two eye crossing times in a front group box eye diagram. This value is then used to compute several vertical eye measurements, such as eye height, eye amplitude, level one, and level zero, etc. If Auto Mode is “off”, the user should enter the time value where he thinks the actual vertical eye measurements should be conducted (at present, only minimum eye height is measured).

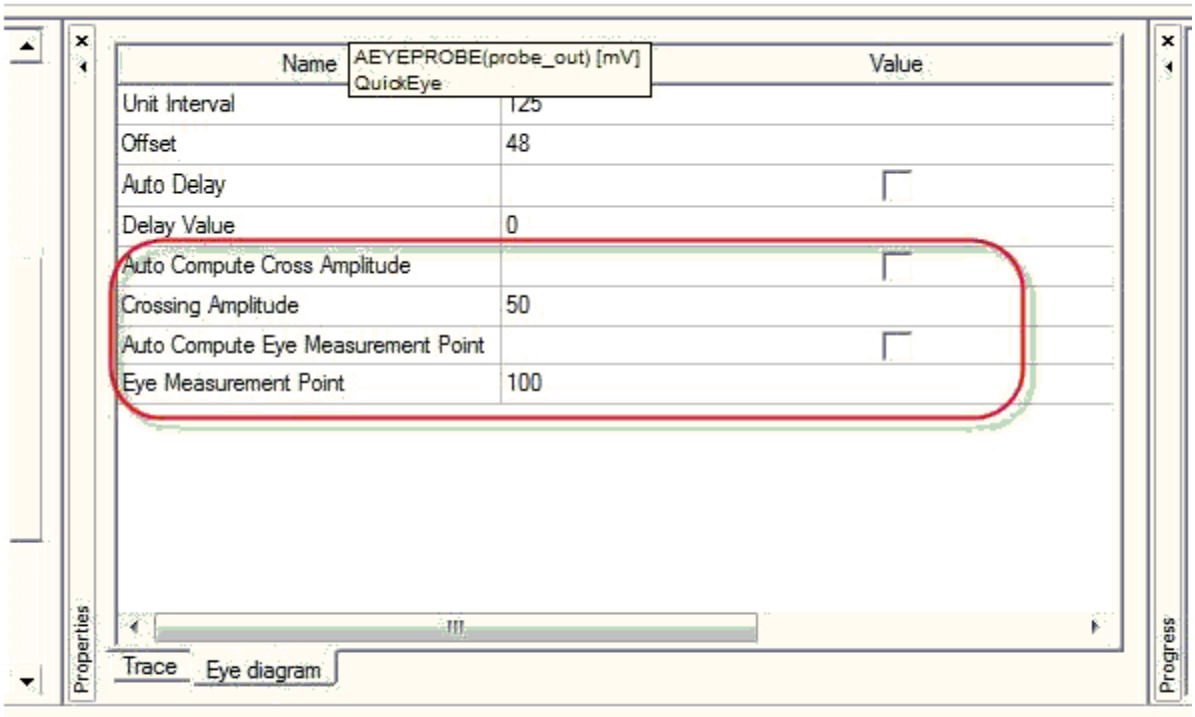
  Auto Delay does not affect this value if the user follows the eye diagram that is currently displayed. Consequently, the user should enter the diagram time value where he thinks the eye measurement point actually lies. For example, let 1ns be the unit interval (UI) and assume the front group box eye is drawn over 2ns. With Auto Delay “off”, if the first crossing is at 0.3ns and the second is at 1.3ns, the user should enter 0.8ns as the approximate eye measurement point. With Auto Delay “on”, the first crossing is shown at 0.5ns and the second is shown at 1.5ns, so the approximate eye measurement point is 1ns.

The eye measurements described above can be accessed by analyzing **QuickEye** simulation in HighSpeedChannel.adsn (Nexxim standard examples) and putting your cursor on **AEyeProbe**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The two check boxes are in the **Eye diagram** tab of the 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.

Eye Measurement Range Function Parameters (2D and Circuit)

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 field for the parameter name. Specify the unit for the parameter by selecting from the Unit drop-down menu.

Name	DefaultValue	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.

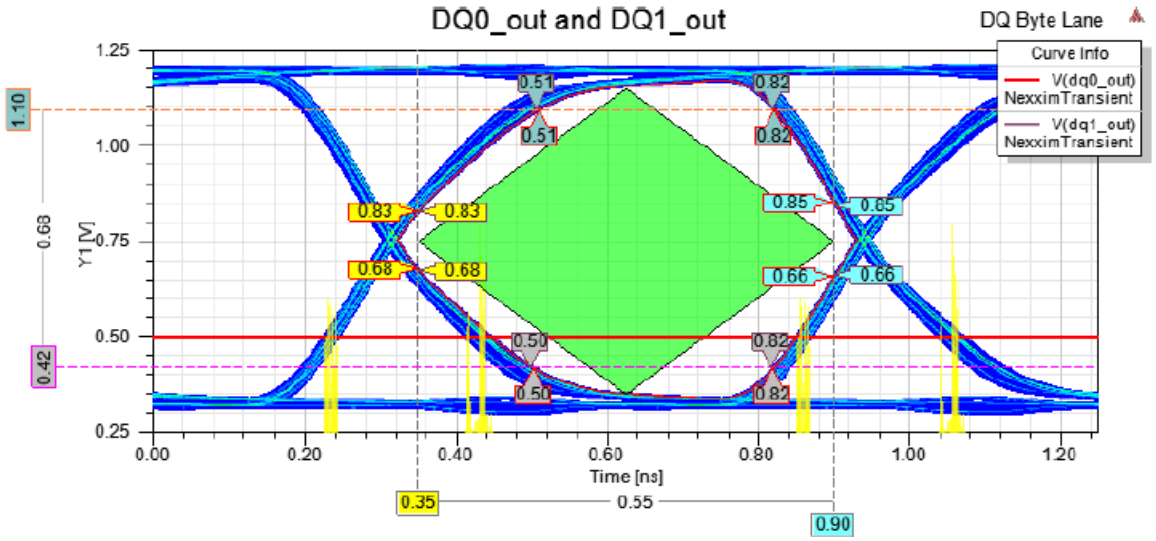
Using Stacked Eye Diagrams

The Rectangular Eye Diagram is optimized for viewing a single eye. But when multiple eyes are overlaid in the same Rectangular Eye Diagram, it is not very easy to compare the eyes. It is also very tedious to compare Y-marker and X-marker intersections. The Stacked Eye Diagram feature allows multiple eye plots to be stacked vertically for easy comparison. For details on the setup, see Stacked Eye Diagram.

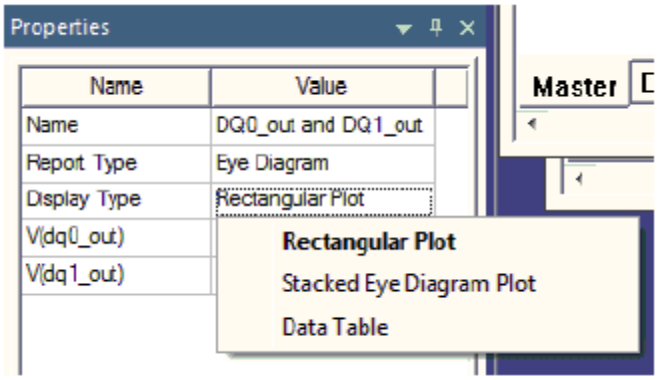
Converting a Rectangular Eye Diagram to Stacked Eye Diagram

The following Rectangular Eye Diagram has two eyes. This is not clearly evident from this plot, except on the legend, and hence the two eyes cannot be compared easily.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

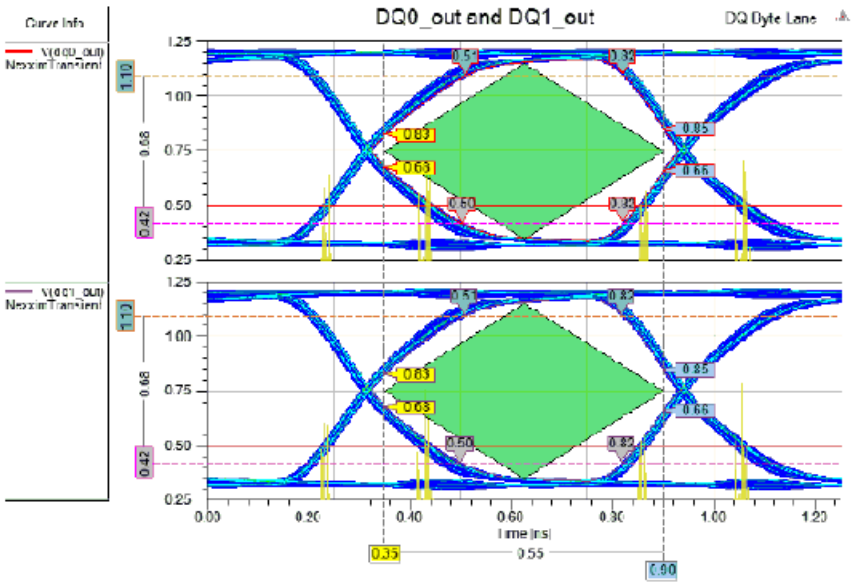


To convert this plot to a Stacked Eye Diagram, select the plot in the Results folder in the **Project Manager** window and change its **Display Type** property to **Stacked Eye Diagram Plot** as shown in the following screenshot.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After conversion, the Eye diagrams are stacked as in the following view:



Only Rectangular Eye Diagram display types can be converted to Stacked Eye Diagrams. Statistical Eye Diagram Plots cannot be converted to Stacked Eye Diagrams.

In a Stacked Eye Diagram all the eyes are stacked over each other and are easily comparable in terms of X and Y data ranges, eye mask violations, histograms, X- and Y-marker intersections.

Histograms

Histograms can be enabled in a Stacked Eye Diagram by selecting any eye , then enabling the Show Histogram property under the Histogram tab in the Properties window, just as in Rectangular Eye Diagrams. The histograms are enabled for all eyes in the Stacked Eye Diagram, and are aligned as shown in the illustration above.

X and Y Markers

X-markers and Y-markers can be added to Stacked Eye diagrams on the **Report 2D > Marker** menu. X-markers align across all the stacked eyes.

Y-markers are specific to each eye. When you set up a Y-marker, select Current Stack to put the Y-marker only on the selected diagram, or All Stacks to add Y-markers to all diagrams. The Y-markers move up and down in tandem.

Eye Masks

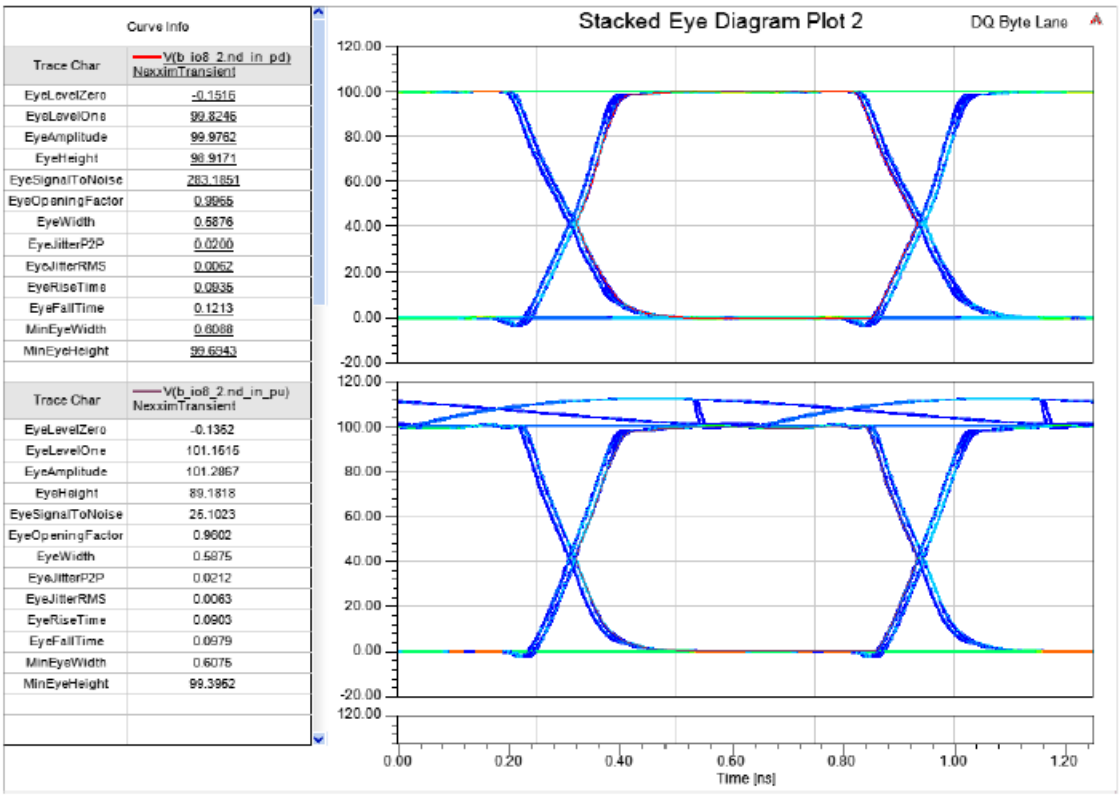
An Eye Mask can be enabled and specified just as in a Rectangular Eye Diagram. If enabled and specified, the same Eye Mask is shown for each eye in Stacked Eye Diagram.

Eye Measurements

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eye measurements can be enabled through the shortcut menu selection **Trace Characteristics > Add All Eye Measurements**.

Eye Measurements are shown in a Legend window. To maximize plot area, Eye Measurements and other trace characteristics are shown in rows instead of columns for Stacked Plots, alongside each stack, as show in the following:



2D and Circuit PAM-4 Eye Plot

Eye diagrams created from PAM-4 voltage traces differ from regular (NRZ) eye diagrams because they have 4 symbol levels that create three distinct sub-eye openings between the three sequential pairs of symbol levels.

A PAM-4 modulation can be set up using any of the following:

- Transient Analyses
- Eye Source for QuickEye
- AMI Transmitter for AMI Analysis

For more information, see PAM-4 Transient Analysis.

After a successful analysis using PAM-4 modulation, to display an eye plot of the Transient, QuickEye, or AMI results:

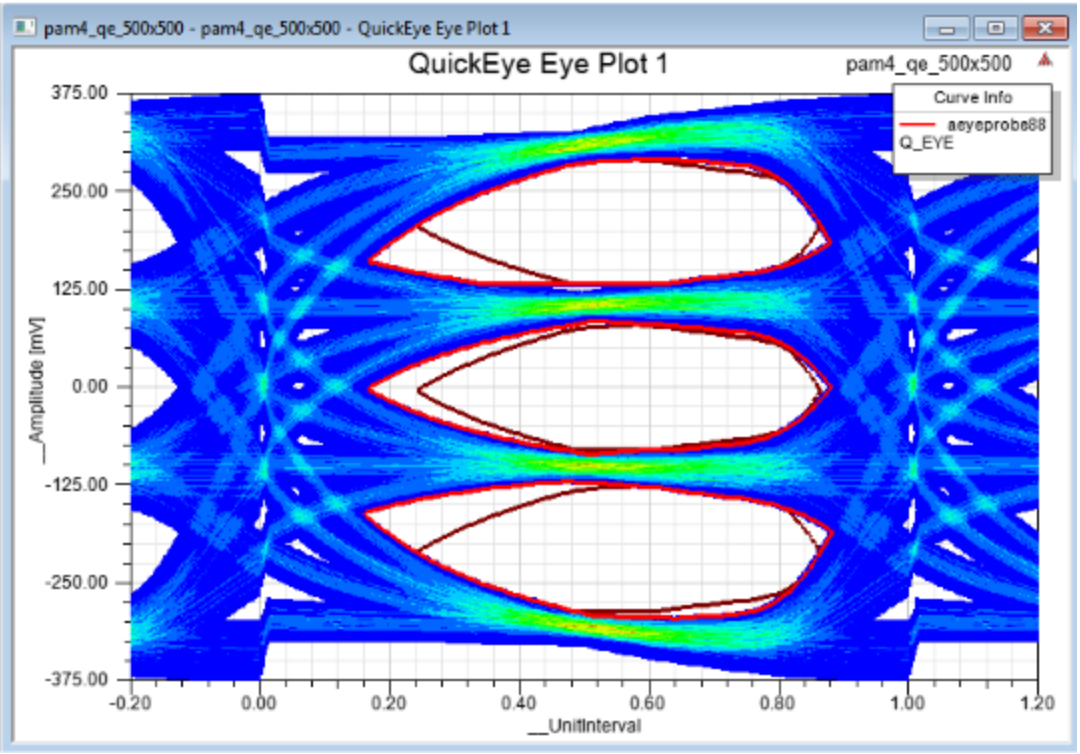
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Product** > **Results** or, on the **Project Manager** window, expand the **Project Tree** and [*active design folder*], then right-click **Results**. Navigate to the appropriate **report type** . Then select **Statistical Eye Plot** or **Eye Diagram Plot** to open the **New Report** window.
2. Selections in the report window are context-sensitive. Whatever Eye Probe/Scope settings are chosen determine the relevant output quantities from which you may select. For example, setting the Eye Scope context to an existing Eye Probe result in the display of quantities that are relevant to the Eye Probe. Whereas choosing Others in the Eye Scope list box displays all other voltage, current and power quantities.

   Select the output quantities of interest and click **Add Trace**.
3. Click **Done**. A Report window opens to display a graph of the analysis results.

Statistical Eye Plots of PAM-4 Results

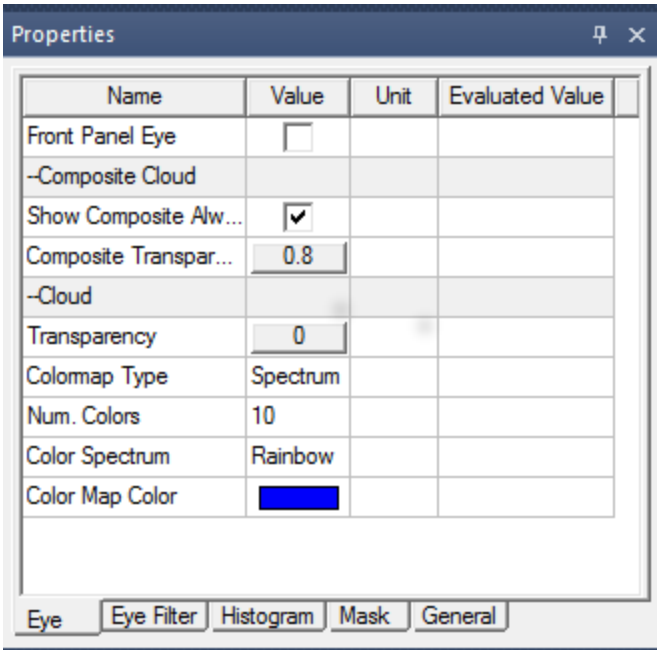
A Statistical Eye Plot of a PAM-4 transmission resembles the following:

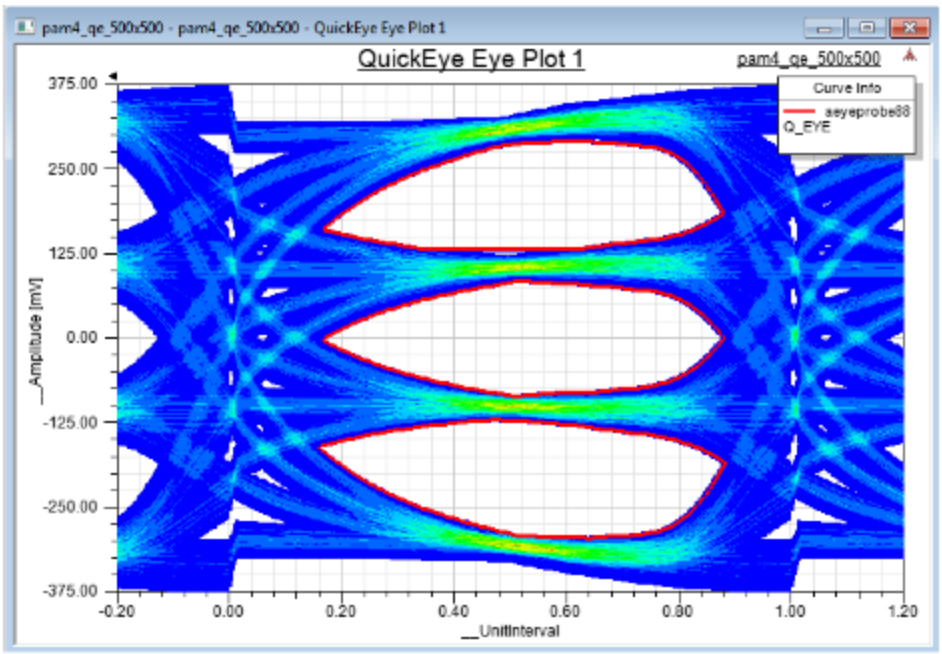


An additional “Consolidated” eye, is also displayed. The consolidated eye is created by superimposition of the three sub-eyes aligned along their voltage reference levels. The consolidated eye shows the worst-case eye opening and serves as a simplified measure of the goodness of the eye. The user chooses to use the visual and measurements on the consolidated or individual sub-eyes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The consolidated eye display is controlled by a trace property:



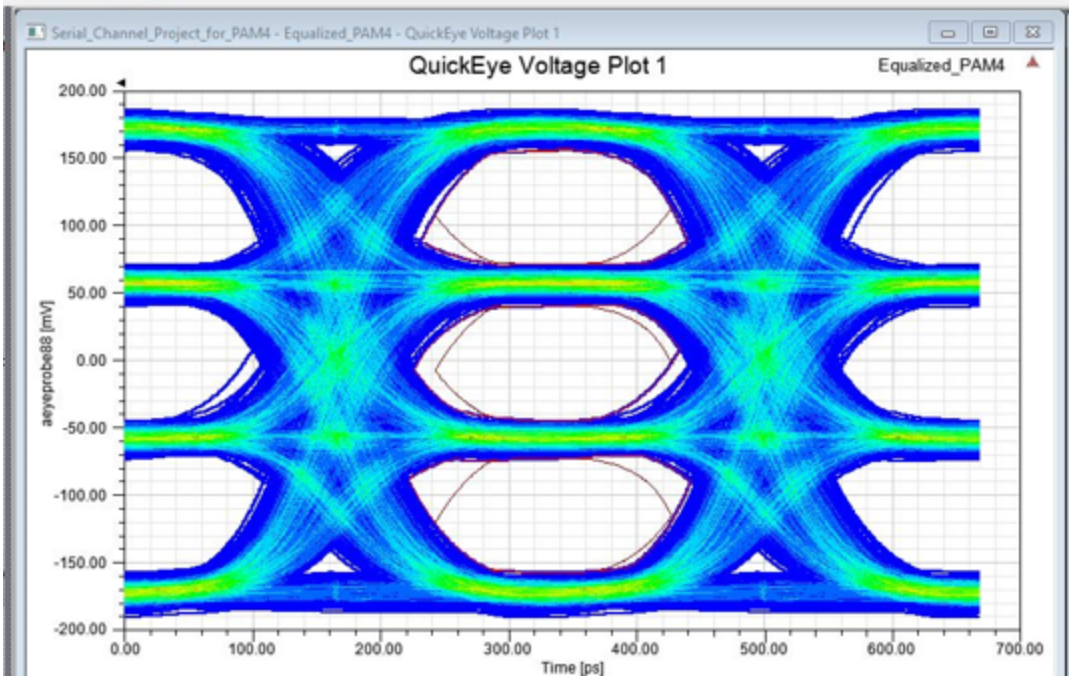


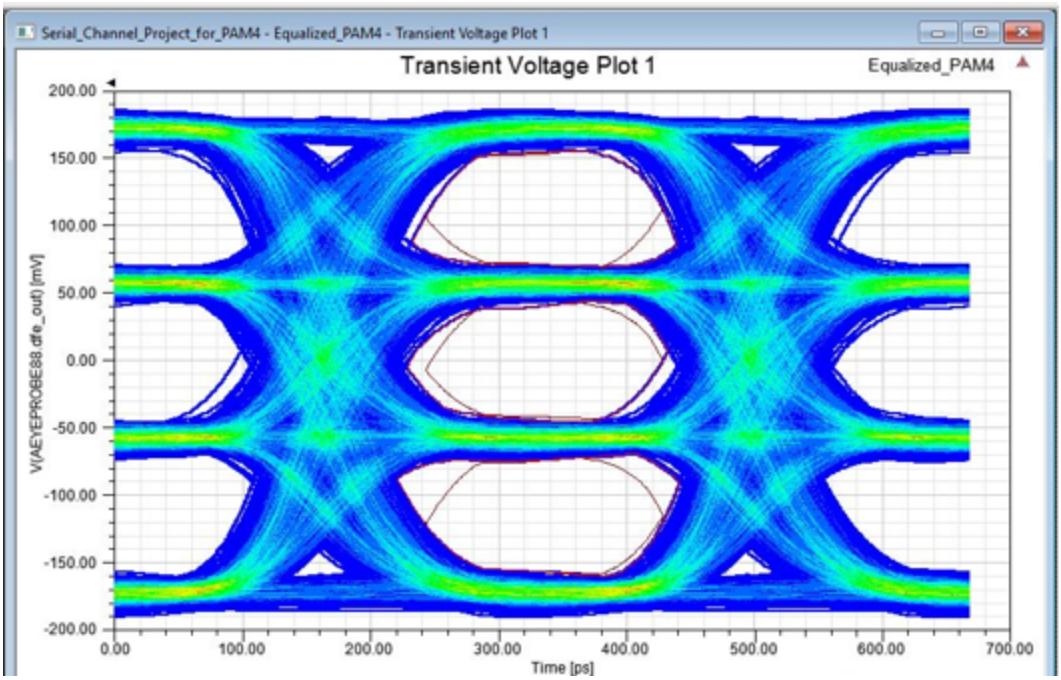
Voltage Plots of PAM-4 Results

When you generate a Rectangular Eye Diagram from PAM-4 data, the waveform sub-view can be zoomed in to show the boundaries of the unit intervals (black lines) and the inner-eye contour

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

overlay:

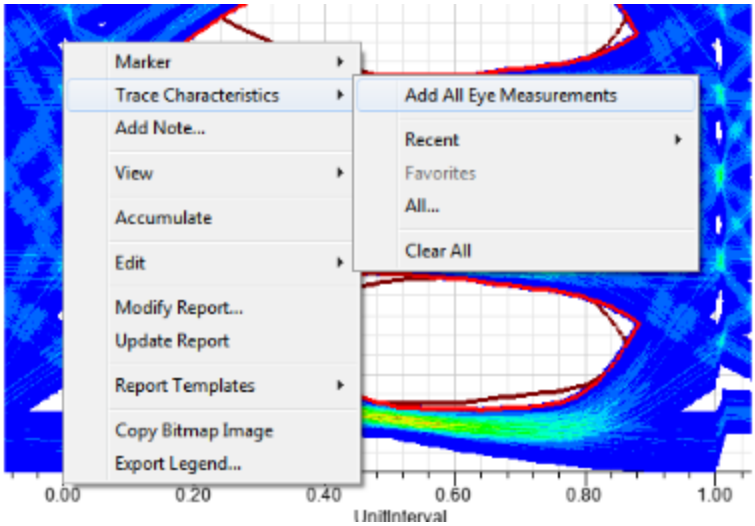




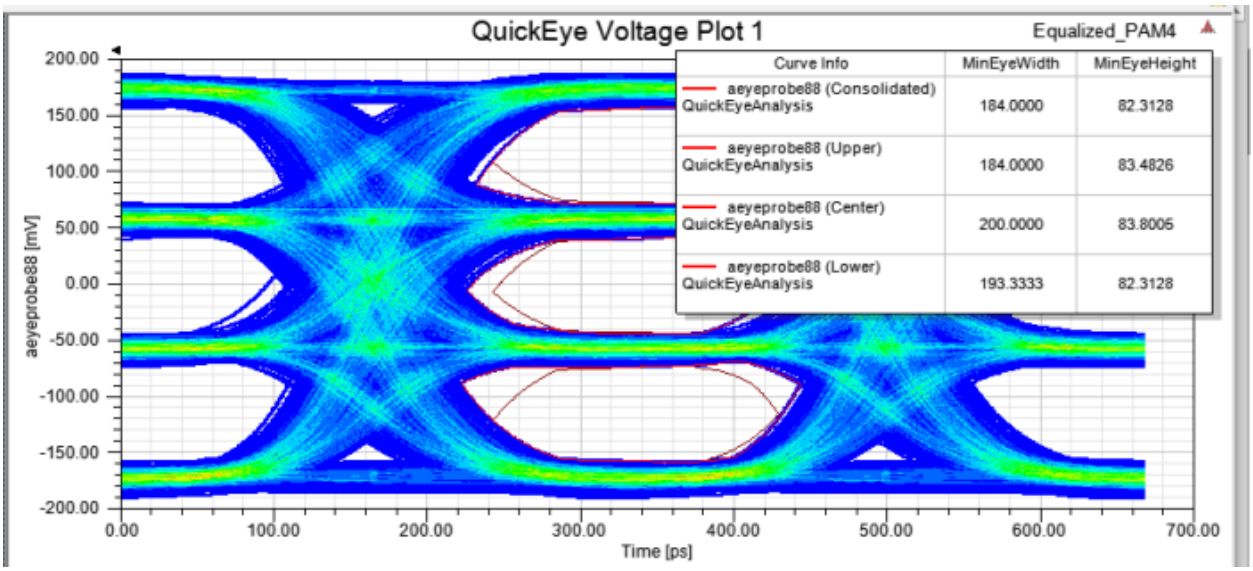
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eye Measurements for PAM-4 Plots

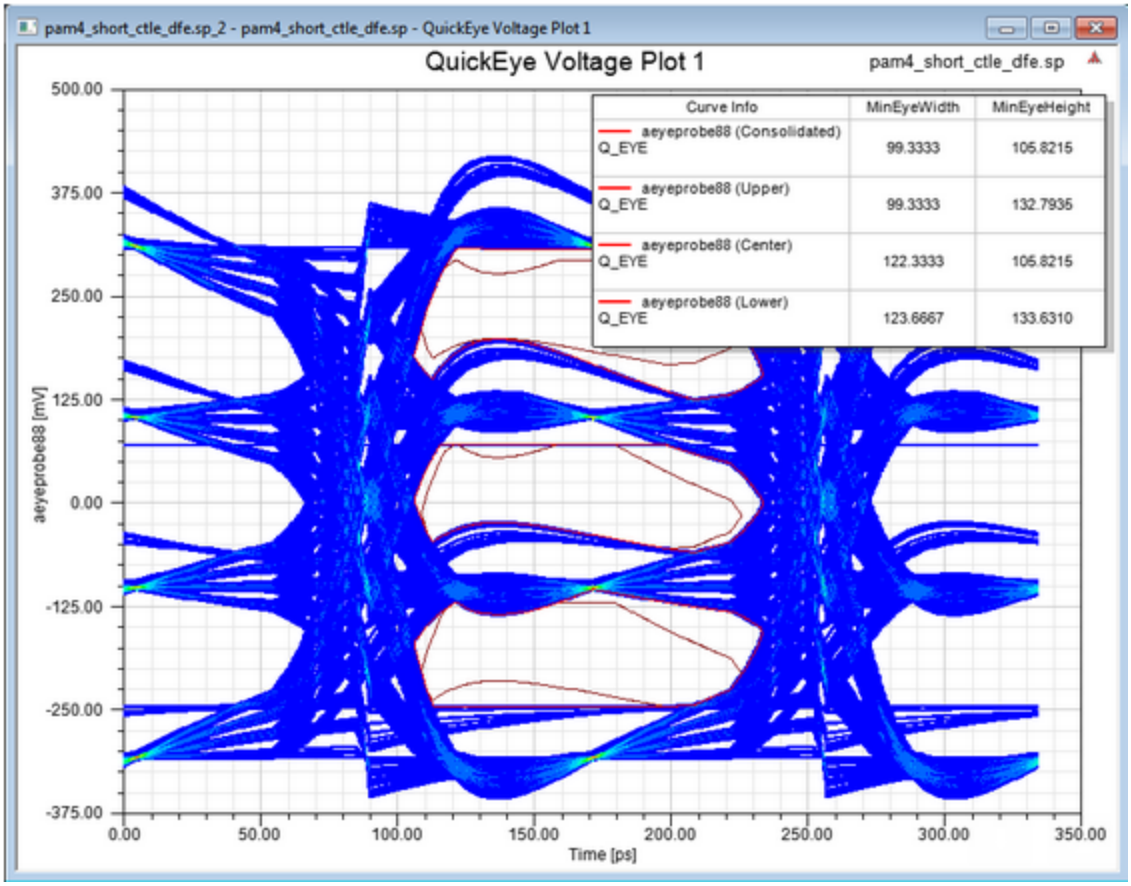
PAM-4 eye diagrams support only the MinEyeWidth and MinEyeHeight measurements. To add trace characteristics, right-click the plot and select on the following options:



When **Add All Eye Measurements** is clicked, only the supported MinEyeHeight and MinEyeWidth are added to the plot. Any other measurements display NaN if added.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



PAM-4 trace characteristics are displayed for each sub-eye and include the consolidated measurement. The sub-eye names and top-to-bottom sorting allow easy mapping of the numbers to the visual. When the Legend is exported (via the shortcut menu), the entire table along with the sub-eye name annotations is also exported.

> **Note:** Mask violations for transient analysis are not supported.

Related Topics

2D and Circuit Report Types

2D and Circuit New Report Window

2D and Circuit Report Display Types

Sweeping a Variable in a Report

Working with Traces

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delta Markers in 2D Plots

Working with Eye Diagrams

Modifying the Background Properties of a Report

Specifying Output Variables in 2D and Circuit Reports

The **Output Variables** window contains four group boxes:

- **Context —** Specify the Report type, the Solution, and for appropriate report types, the Domain and Show fields. Changing the Report type affects whether the Domain menu appears, and may affect the functions listed in the **Function** group box.
- **Output Variables —** Specify the name and expression for a new output variable.
- **Calculation —** Insert quantities into the **Expression** area of the **Output Variables** group box.
- **Function —** Insert completed expressions into the **Expression** field of the **Output Variables** group box.

2D and Circuit Adding a New Output Variable

To add an output variable:

1. Click **Product Menu** > **Results** > **Output Variables** or, in the **Project Manager** window, right-click **Results** and select **Output Variables** to open the **Output Variables** window. Variables defined using the **Product Menu** > **Results** > **Output Variables** command appear in the list at the top of the window.
2. In the **Output Variables** group box, enter a name for the new variable in the *Name* field.
3. To enter an expression, do one or both of the following:
    a. Type part or all the expression directly in the **Expression** field. Valid functions appear in blue. Invalid functions appear in red.
    b. Insert part or all the expression using the options in the Function group box.
4. Click **Add** to add the new variable to the list.
5. Repeat steps 2 through 5 to add additional variables.
6. When you are finished adding output variables, click **Done** to close the **Output Variables** window.

2D and Circuit Building an Expression Using Existing Quantities

When entering an expression for a new output variable, insert part or all the expression using the options in the **Function** group box in the **Output Variables** window.

To add an input variable by inserting part or all of 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.

From **Product Menu** > **Results** > **Output Variables** or, in the **Project Manager** window, right-click **Results** and select **Output Variables** to open the **Output Variables** window. In the **Output Variables** window, enter a name for the new variable in the *Name* field.

To insert a quantity:

1. From the **Report Type** drop-down list, select the type of report from which the user wants to select the quantity.
2. From the **Solution** drop-down menu, select the solution from which the user wants to select the quantity.
3. From the **Category** group box, select the appropriate type of quantity to enter.
4. From the **Quantity** group box, select the appropriate quantity or the geometry.
5. From the **Function** group box, select a ready-made function (this option is the same as inserting the function on the **Function** group box).
6. If applicable, on the **Domain** list, select the solution domain.
7. Click **Insert Into Expression**.

   The selected quantity is entered into the **Expression** area of the **Output Variables** window.

To insert a function:

1. From the **Function** group box, select a ready-made function on the drop-down menu.
2. Click **Insert Function into Expression.**

   The function appears in the **Expression** area of the **Output Variables** group box.

1. When the user is finished defining the variable in the **Expression** field, click **Add** to add the new variable to the list.
2. Repeat steps 2 through 6 to add additional variables.
3. When the user is finished adding output variables, click **Done** to close the **Output Variables** 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: Remember that the evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, the user can plot values in units other than SI. If the user wants to plot the polar angle of a complex simulation result, S_{11} say, 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(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 the user treats "1" as degrees, make it explicit and use "1deg + ang_deg(S_{11})" instead.

If the user 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 so by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val(S_{11}) is 90 and "1 + ang_deg_val(S_{11})" is 91.

Deleting Output Variables (2D and Circuit)

To delete output variables:

1. Remove all references to the output variable in the project.
2. Save the project to erase the command history.
3. Click **Product Menu > Results > Output Variables** or, in the **Project Manager** window, right-click **Results** and select **Output Variables**.

This opens the **Output Variables** window.

4. Select the variable and click **Delete**.
5. Click **OK** to close the window.

Viewing Matrix Data in 2D and Circuit Reports

To view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution:

1. From the **Project Manager** window, right-click the solution setup of interest, then click **Matrix Data** to open the **Solution Data** window

The **Matrix Data**tab is selected.

2. In the **Design Variation** field, specify the design with the matrices you want to view.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optionally, choose a design variation solved during an Optimetrics analysis on the **Set Design Variation** window. This lists all the solved variations in the design.

3. In the **Simulation** dropdown menu, select the solution setup and solved pass - adaptive, single frequency solution, or frequency sweep - for which you want to view matrices.
4. Select the type of matrix you want to view: **S-matrix**, **Y-matrix**, **Z-matrix**, **Gamma**, or **Z_o** (characteristic impedance.) The available types depend on the solution type.
5. Select the format in which to display the matrix information. The available formats depend on the matrix type being displayed.
6. Select the solved frequencies to display:

- to open the matrix entries for all solved frequencies, select **All Freqs**. It is selected by default.
- To show the matrix entries for one solved frequency, clear **All Frequencies**, then select the solved frequency for which you want to view matrix entries. For adaptive passes, only the solution frequency specified in the **Solution Setup** window is available. For frequency sweeps, the entire frequency range is available.
- To insert or delete one or more displayed frequencies, click **Edit Freqs**. This command is only available if the sweep type is Fast or Interpolating. If choose to export the matrix data for the Fast or Interpolating sweep after modifying the frequencies in the **Edit Frequencies** window, only those frequencies displayed under the **Matrix Data** tab is exported.

The data is displayed in the table. By default, wave ports are listed in alphabetical, then numerical order, just as they appear in the excitation tree. To change the port order, change setting for Default Matrix sort order in **Electronics Desktop** General options.

2D and Circuit Selecting the Matrix Display Format

Display matrix data in the following formats. The available formats depend on the type of matrix being displayed.

Magnitude, Phase (deg)	Displays the magnitude and phase (in degrees) of the matrix type.
Real, Imaginary	Displays the real and imaginary parts of the matrix type.
dB, Phase (deg)	Displays the magnitude in decibels and phase in degrees of the matrix type.
Magnitude	Displays the magnitude of the matrix type.
Phase (deg)	Displays the phase in degrees of the matrix type.
Real	Displays the real parts of the matrix type.
Imaginary	Displays the imaginary parts of the matrix type.
dB	Displays the magnitude in decibels of the matrix type.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Renaming 2D and Circuit Matrix Data

From the **Project Manager** window, right-click a port excitation to rename it. When you rename a port excitation, the associated data is reordered so it can be presented in the same manner. The reordering is done to match the tree-sort order presented for the ports (renamed matrix data is reordered so alphabetic values appear before numeric values).

Exports of the matrix data is ordered in the same manner. This reordering is conducted as part of post-processing and does not force a re-solve.

Animation in 2D and Circuit Reports

The following topics describe how to post-process field overlay displays to create and view various animated reports. Any of the field overlay displays can be animated by cycling the overlays as a series of frames. See the following topics for details on creating the field overlays:

- Overlaying Surface Currents on a 3D View
- Overlaying Far Fields on a 3D View
- Overlaying Near Fields on a 3D View

There are two modes of animation: frequency-based and phase-based. Frequency-based animation creates a series of frames by calculating the field overlay at each of the swept frequencies. Phase-based animation creates a series of frames by calculating the field overlay at a fixed set of phase deviations around a selected frequency. The animation controls can also change the simulation design point.

Related Topics

2D Frequency Animation

2D Phase Animation

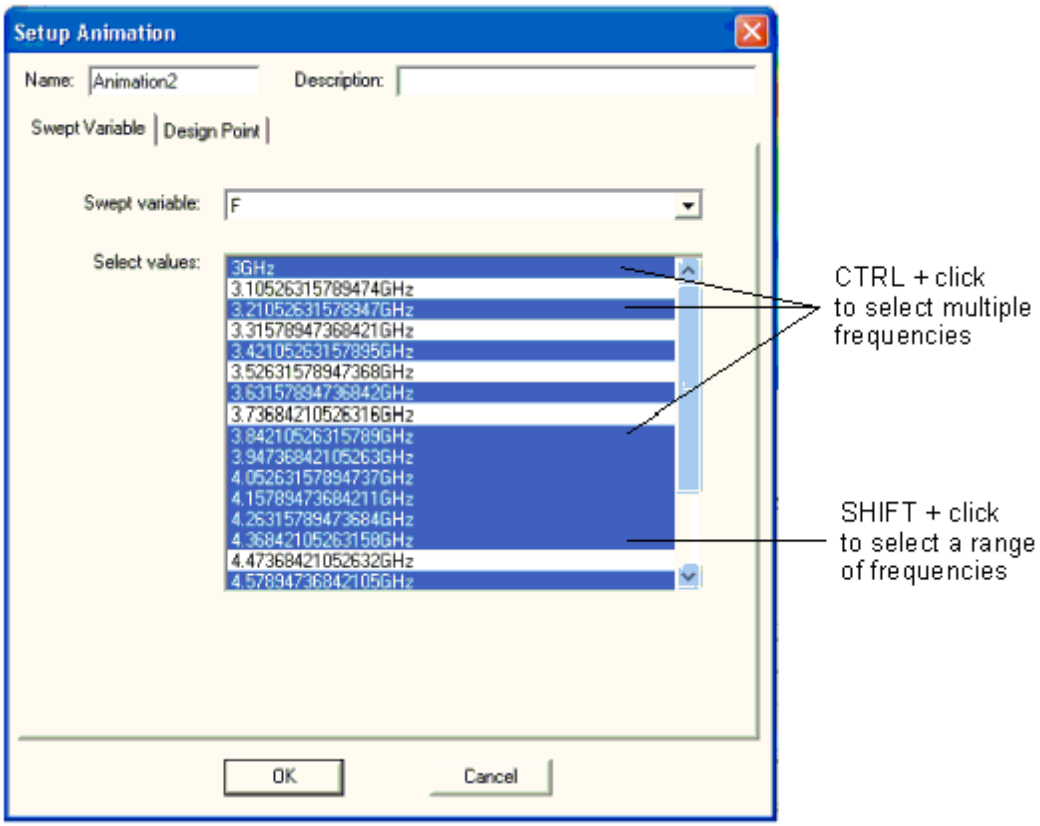
Changing the Design Point in 2D Animation

2D Frequency Animation

1. To initiate the animation of the overlay that is currently displayed, do one of the following:
   - Select **Animate** on the **View** menu.
   - Expand the **Results** icon in the Project window, right-click the overlay entry, and select **Animate** from 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.

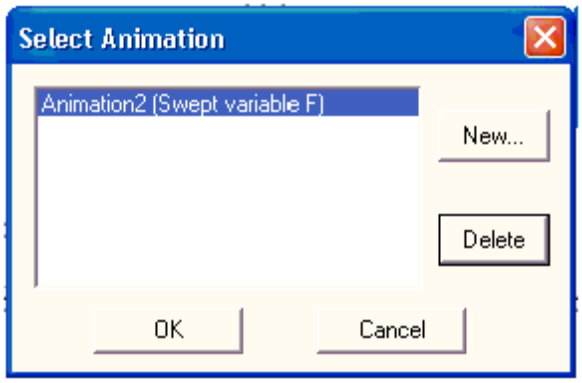
2. If no animations have been defined previously, the **Setup Animation** window opens:



- Specify a name in the **Name** field (or accept the default, **Animation***n*, where *n* is a numeral). Optionally, enter a description.
- For a frequency animation, select **F** as the **Swept Variable**.
- By default, all the frequencies are selected (highlighted). Hold down **Ctrl** to select multiple individual frequencies, or hold down **Shift** to select a contiguous range of frequencies. [These selection modes are illustrated in the window example above].
- Click **OK**.

If one or more animations have been defined, selecting **Animate** from one of the menus opens the **Select Animation** window:

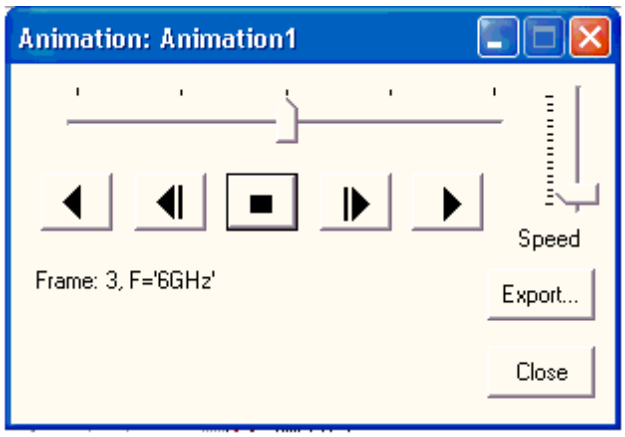
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Select Animation** window provides the following operations:

- Click to select one of the animations and click **OK** to start that animation.
- Click **New** to open the **Setup Animation** window described above, and close the **Select Animation** window.
- Select an animation and click **Delete** to delete the definition.

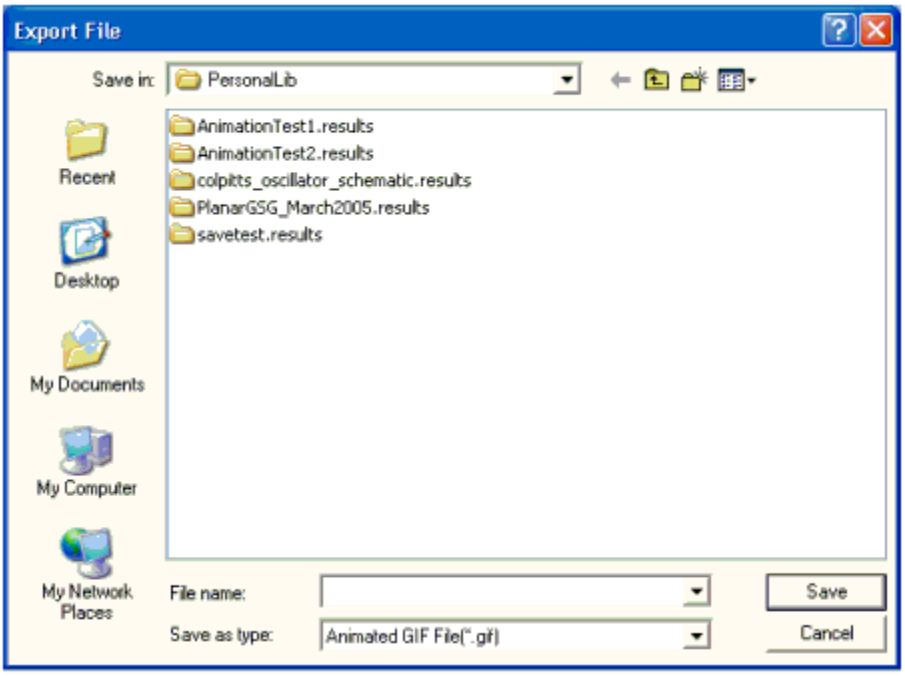
3. Electronics Desktop calculates the frame data. If the Progress window is displayed, monitor the progress of the calculation. When the frames have been calculated, the animation begins and the **Animation** control window opens:



- Click the playback buttons to play the animation. From left to right, the buttons are **Reverse**, **FastReverse**, **Stop**, **Fast Forward**, and **Forward**. The indicator at the top of the window shows the progress of the animation. Click the **Speed** slider to control the speed of 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.

- To export the frame data to a file, click **Export**. The **Export File** window opens:



Specify the directory and file name. Click the **Save as type** menu to select the file format (Animated GIF or AVI). Click **Save** to save the data and close the window.

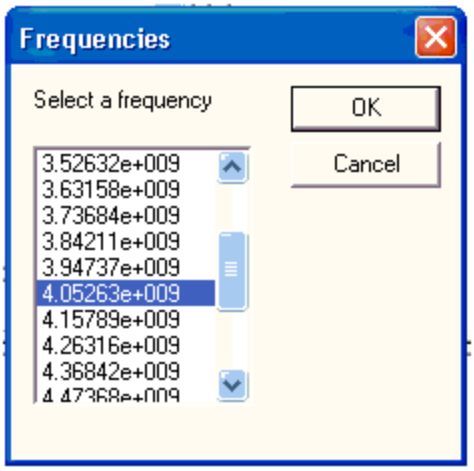
- Click **Close** on the **Animation** control window to stop the animation and close the window.

2D Phase Animation

1. To prepare for phase animation, you must select the base frequency.
- For surface current and far field overlays, expand the **Results** icon in the **Project Manager** window, right-click the overlay, and select **Frequency** on the menu. The

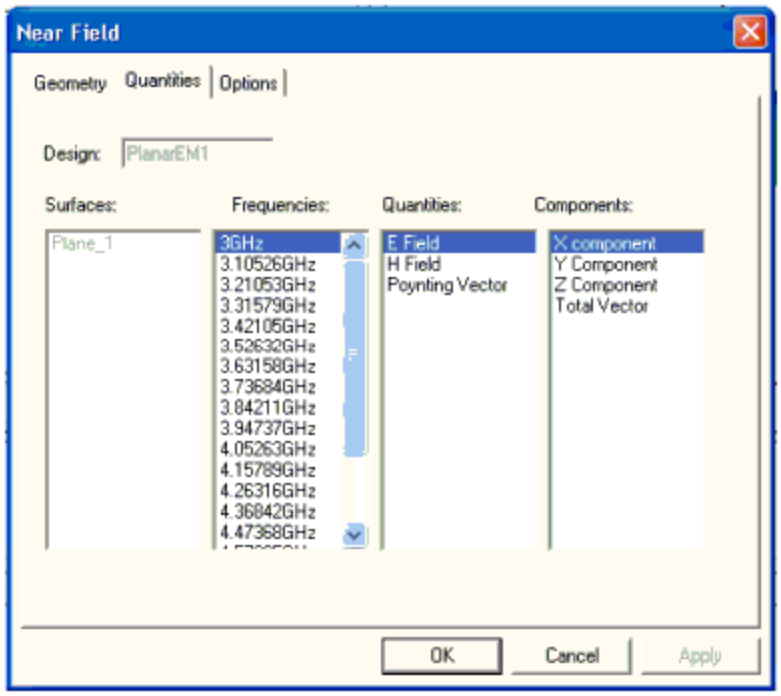
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequencies window opens:



The list displays the frequencies that are swept in the analysis. Select a frequency on the list; the overlay displays the field values calculated at that frequency. Click **OK** to leave the overlay at the selected frequency.

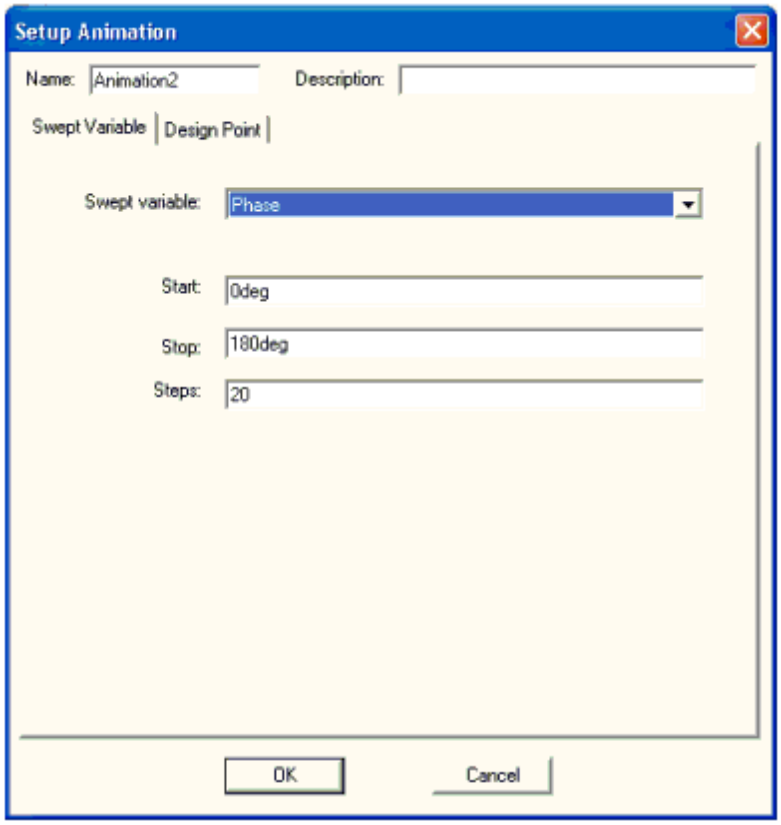
- For near field overlays, expand the **Results** icon in the **Project Manager** window, right-click the overlay, and select **Properties** on the menu. Click the **Quantities** tab to select a reference frequency for the phase animation:



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 apply the frequency and close the window.

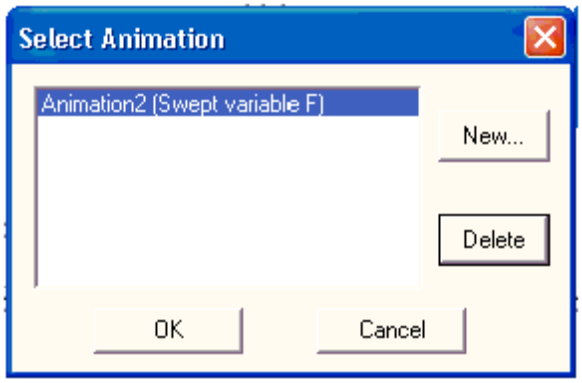
2. To initiate the animation of the overlay that is currently displayed, do one of the following:
   - Select **Animate** on the **View** menu.
   - Expand the **Results** icon in the Project window, right-click the overlay entry, and select **Animate** from the menu.
3. If no animations have been defined previously, the **Setup Animation** window opens:



- Specify a name in the **Name** field (or accept the default, **Animation***n*, where *n* is a numeral). Optionally, enter a description.
- For a frequency animation, select **Phase** as the **Swept Variable**.
- Select the Start and Stop phases in degrees.
- Click **OK**.

If one or more animations have been defined, selecting **Animate** from one of the menus opens the **Select Animation** window:

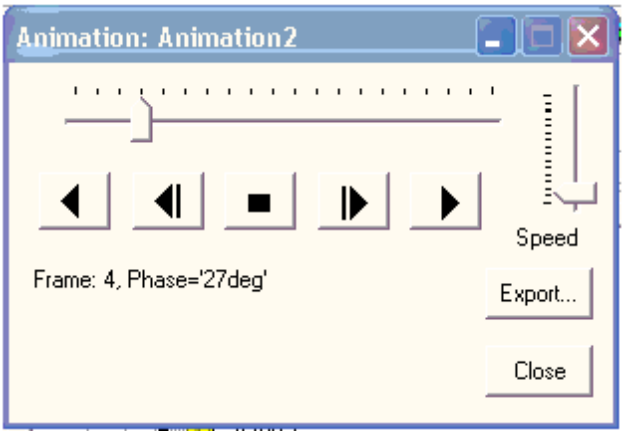
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Select Animation** window provides the following operations:

- Click to select one of the animations and click **OK** to start that animation.
- Click **New** to open the **Setup Animation** window described above, and close the **Select Animation** window.
- Select an animation and click **Delete** to delete the definition.

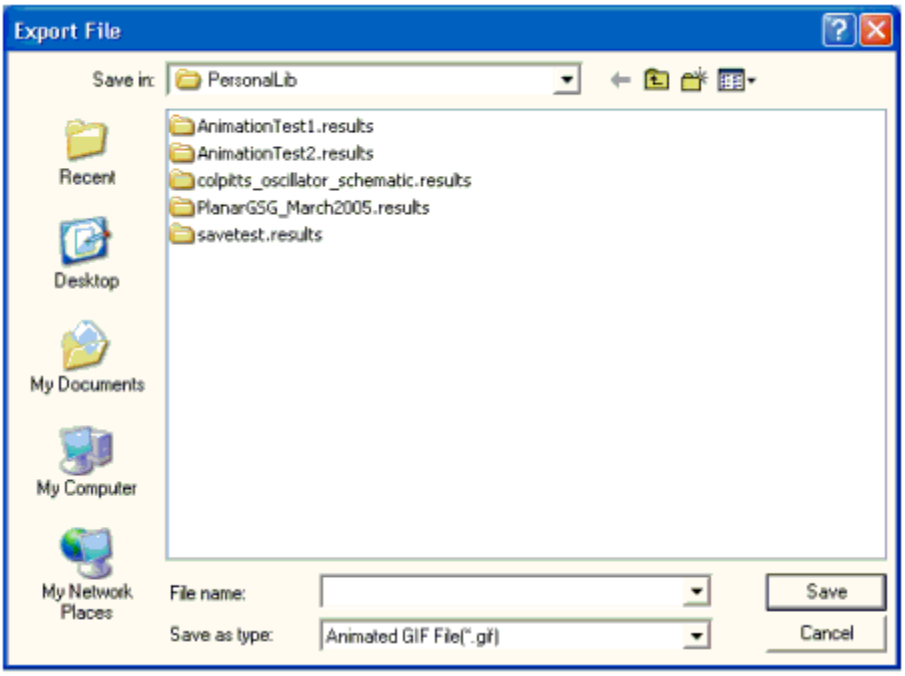
4. Electronics Desktop calculates the frame data. If the Progress window is displayed, monitor the progress of the calculation. When the frames have been calculated, the animation begins and the **Animation** control window opens:



- Click the playback buttons to play the animation. From left to right, the buttons are **Reverse**, **FastReverse**, **Stop**, **Fast Forward**, and **Forward**. The indicator at the top of the window shows the progress of the animation. Click the **Speed** slider to control the speed of 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.

- To export the frame data to a file, click **Export**. The **Export File** window opens:



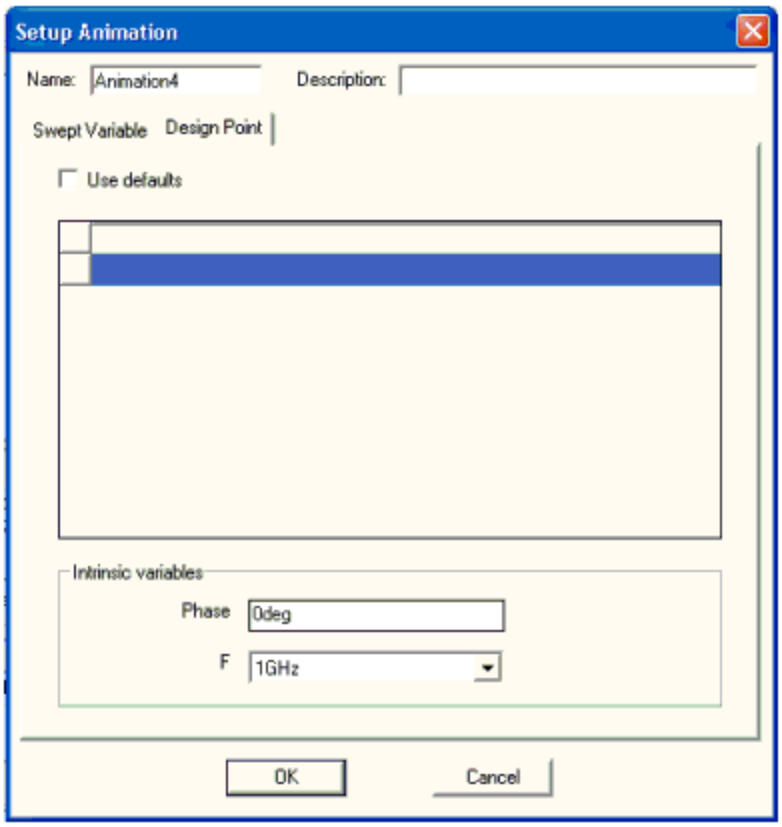
Specify the directory and file name. Click the **Save as type** menu to select the file format (Animated GIF or AVI). Click **Save** to save the data and close the window.

- Click **Close** on the **Animation** control window to stop the animation and close the window.

Changing the 2D Design Point

Calculating frames for an animation is equivalent to re-simulating the planar design. You can specify design point parameters for the animation calculations that are different on the ones used in the original simulation. From the **Animation Setup** window, select the **Design Point** tab and deselect the **Use defaults** option. The following fields 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.



Make any appropriate changes, then click **OK** to apply the changes and close the window. Clicking **Cancel** closes the window without making any changes.

User-Defined Outputs in 2D and Circuit Reports: Introduction

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 of **syslib**, **userlib** or **Personallib** with any directory structure needed for organization (the **Lib** directory name is special and its purpose is explained later on in the document)

The UDO scripts that are 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Named Probes and Properties in user-defined Outputs

Computation of Traces Based UDO Calculations

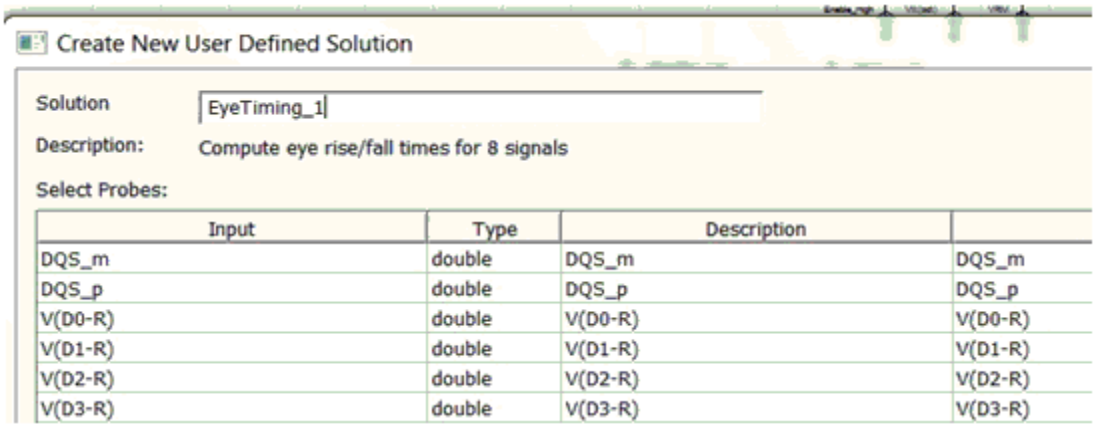
Dimensions Reduction by UDO Calculations

Dynamic Probes

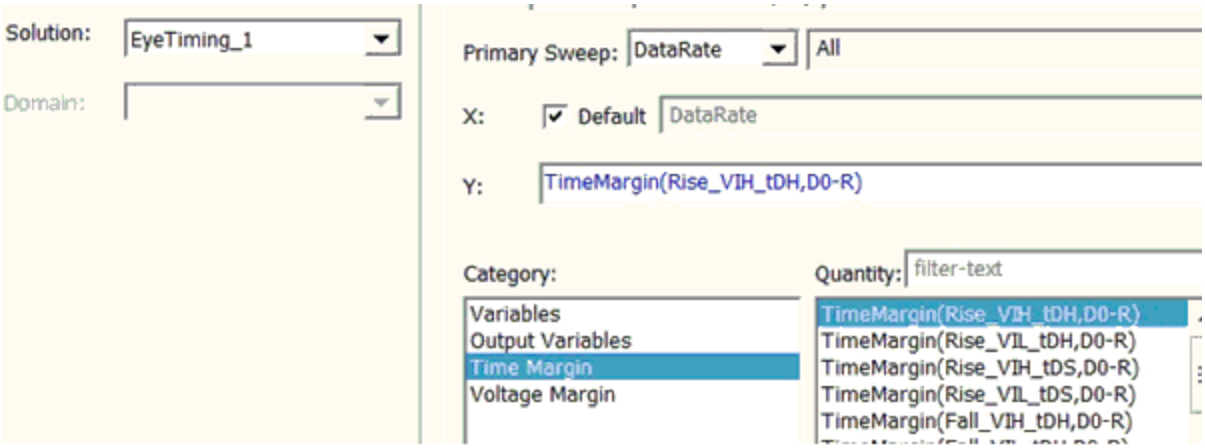
Named Probes and Properties in user-defined Outputs (2D and Circuit)

UDOs allow processing data across traces, solutions and report types. A UDO specifies the named probes and properties for which user selects/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/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.



You can then access this solution in the Reporter.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Computation of Traces Based UDO Calculations (2D and Circuit)

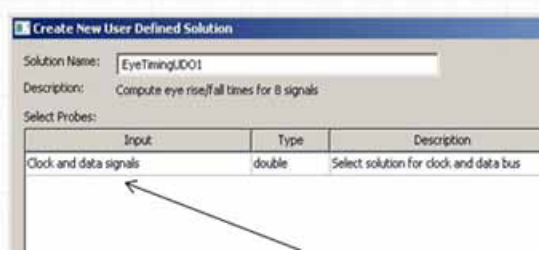
When traces that are based upon UDO outputs are computed, the data for probes is computed and passed to the UDO script for each design variation. Along with the probe data, the values of properties entered by user are also passed. The information about the UDO calculations that need to be computed is also made available. The UDO then performs the computation and passes the results to reporter. Note that UDOs can compute and pass back more calculations than have been requested at that point of time. This allows UDOs to compute a set of calculations that take almost same amount of computational resources as any one calculation in that set and cache that with reporter. *When those calculations are subsequently plotted by user, reporter uses the cached results instead of invoking the computation on UDO*.

Dimensions Reduction by UDO Calculations (2D and Circuit)

The probes in a UDO can have heterogeneous dimensions of data (e.g., one probe in a UDO can have data that is 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* potentially being different). UDOs allow reducing any number of these intrinsic variables. In the previous example UDO calculations can be 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 (2D and Circuit)

UDOs can also 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 having 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 end user to 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, and information about solution, context and intrinsic variables is used from user selected probe, however multiple dynamic probes can be associated with same user selected probe. The dynamic probes are sampled from UDOs at the time of trace computation and not at the time of creation of user-defined solution.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This means that you select solution, context, 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 on the UDO code.

user-defined Outputs: Python Script API (2D and Circuit)

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

Optional Functions in IDO Extension Abstract Class

Data Types Used in Python Script

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 (2D and Circuit)

The purpose, argument list and expected return types for each of the **IUDOPluginExtension** abstract methods, which the UDO author is expected to implement are shown in the following diagram.

- Import Statements
- UDOExtension Class
- IUDOPluginExtension Abstract Class

Import Statements (2D and Circuit)

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 UDO script: the following import statements should be added to the top of the python script:

```
from Ansys.Ansys.ModulePluginDotNet.Common.API import *
from Ansys.Ansys.ModulePluginDotNet.Common.API.Interfaces import *
from Ansys.Ansys.ModulePluginDotNet.UDO.API.Interfaces import *
from Ansys.Ansys.ModulePluginDotNet.UDO.API.Data import *
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional Functions in IDO Extension Abstract Class (2D and Circuit)

The following functions, while a part of the IUDOExtension abstract class, have meaningful default implementations and are therefore optional. However, they can be overridden to take advantage of advanced functionality.

Validate(List<string> errorStringList,

Data Types Used in Python Script (2D and Circuit)

There are several types that you must use while authoring the python script. Some of them are used to pass data from UI to python script and provide interface for working with this data. Some are used to pass data from python script to UI.

To pass data from python script to UI the objects of the C# class must be created in python script using their C# constructors. The objects can then be set as functions return values or set to the output parameters using their API.

Working With Properties for UDO (2d and Circuit)

A property is the unit for collecting and using input on the user that is used to influence the UDO's Compute method. These are initially set up when the UDOs **GetInputUDSParams** method is called and are retrieved in the compute method.

There are 3 supported property types that could be used in the UDO script:

- INumberProperty to specify number properties (with unit support).
- IMenuProperty to allow the user to select from a list of options.
- ITextProperty to allow the user to enter text.

The IPropertyList type implements a collection for these properties.

IPropertyList Abstract class

IProperty Abstract class

INumberProperty Abstract class

ITextProperty Abstract class

IMenuProperty Abstract class

Other Application Specific Classes Used in Python Scripts (2D and Circuit)

This topic describes other classes used 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.

Other Application Specific Classes Used in Python Scripts (2D and Circuit)

This topic describes other classes used in Python scripts:

- Constants Class
- UDSProbeParams Class
- UDSDynamicProbes Class
- QuantityInfo Class
- IProgressMonitor Abstract Class

Constants Class (2D and Circuit)

The constants used in 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
```

Other Application Specific Classes Used in Python Scripts (2D and Circuit)

This topic describes other classes used in Python scripts:

- Constants Class
- UDSProbeParams Class
- UDSDynamicProbes Class
- QuantityInfo Class
- IProgressMonitor Abstract Class

Constants Class (2D and Circuit)

The constants used in python script are defined in the Constants class.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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
```

Other Application Specific Classes Used in Python Scripts (2D and Circuit)

This topic describes other classes used in Python scripts:

- Constants Class
- UDSProbeParams Class
- UDSDynamicProbes Class
- QuantityInfo Class
- IProgressMonitor Abstract Class

Constants Class (2D and Circuit)

The constants used in 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
```


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 (2D and Circuit)

This topic describes other classes used in Python scripts:

- Constants Class
- UDSProbeParams Class
- UDSDynamicProbes Class
- QuantityInfo Class
- IProgressMonitor Abstract Class

Constants Class (2D and Circuit)

The constants used in 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
```

Other Application Specific Classes Used in Python Scripts (2D and Circuit)

This topic describes other classes used in Python scripts:

- Constants Class
- UDSProbeParams Class
- UDSDynamicProbes Class
- QuantityInfo Class
- IProgressMonitor Abstract Class

Constants Class (2D and Circuit)

The constants used in python script are defined in the Constants class.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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
```

Using .NET Collection Classes and Interfaces in Python Scripts (2D and Circuit)

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 topic describes how to work with the .Net collection objects in python scripts.

.NET Array, IEnumerable, and IList objects can be indexed and iterated over as if they are Python lists. You can also check for membership using 'in'. To get .Net Array and IList sizes use python's 'len' or .Net 'Count'.

Example:

Getting size:

```
arraySize = doubleDataArray.Count

arraySize = len(doubleDataArray)


listSize = sweepsNamesList.Count

listSize = len(sweepsNamesList)
```

Iterating:

```
for sweep in sweepsNamesList:
        print sweep

for in xrange(listSize)
        print sweepsNamesList[i]
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Checking for membership:

```
if 'Time' in sweepsNamesList:
        doThis()
else:
        doThat()
```

For .NET IDictionary, 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 returns keys, .Net Dictionary returns .Net KeyValuePair.

Example:

Iterating:

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:
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        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 and time-consuming.

Example:

```
aPythonList = list(dotNetArray)

aPythonList = list(dotNetList)

aPythonDict = dict(dotNetDictionary)
```

user-defined Outputs: Messaging Methods (2D and Circuit)

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 Manager** window using the appropriate icon.

These functions can also be used for debugging purposes.

- **AddErrorMessage(string)**: Call this method to convey an error condition to the user.
- **AddWarningMessage(string)**: Call this method to convey a warning message: typically used for conditions that are not ideal but can be tolerated by the script.
- **AddInfoMessage(string)**: Call this method to convey an informational message to the user. This is the call to use when outputting messages for debugging purposes.

```
##########################################################################
# Imports
##########################################################################
from Ansys.Ansys.ModulePluginDotNet.Common.API import *
from Ansys.Ansys.ModulePluginDotNet.Common.API.Interfaces import *
from Ansys.Ansys.ModulePluginDotNet.UDO.API.Interfaces import *
from Ansys.Ansys.ModulePluginDotNet.UDO.API.Data import *
class UDOExtension(IUDOPluginExtension):
    def __init__(self):
        pass
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
#--- IDA IUDOPluginExtension -----------------------
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 are simple doubles
    # leave them unitless
    return QuantityInfo(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.

```
#--- 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. Only one double quantity is needed
    param1 = UDSProbeParams("probe1",
            "double quantity probe",
            Constants.kDoubleParamStr,
            "", "")
    udsParams.Add(param1)

    # Add the properties the user must supply
    # In this case, ask for a start/end range for
    # X parameters. Since defaults cannot be reasonably provided
    # as the sweep limits are unknown,
    # ask if the limits can be activated.
    prop = propList.AddNumberProperty("X Min", "0")
    prop.Description = "Start X value to consider"

    prop = propList.AddNumberProperty("X Max", "1")
    prop.Description = "End X value to consider"

    # For menus, the first option is the default.
    prop = propList.AddMenuProperty("Activate X Limits", ["No", "Yes"])
    prop.Description = "Activate X range"
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    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
 # There are no sweeps because they have been reduced.
def GetUDSSweepNames(self):
    return []



 #-------------------------------------------------------------------------
# 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 was 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
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        probeData = inData.GetDoubleProbeData("probe1")
        sweepData = inData.GetSweepsDataForProbe(sweeps[0], "probe1")

# 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 computing
# all the quantities.
        minVal = 0
        maxVal = 0
        avgVal = 0

# Check if a range computation needs to be performed
if useXRangeProp == "Yes":
seenAny = False
avgSum = 0
count = 0

# zip is used since sweep data must also be pulled in
# an index and the array notation could also have been used
for probeVal, sweepVal in zip(probeData, sweepData):
if sweepVal < xRangeStart or sweepVal > xRangeEnd:
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
pass

# Note that in a better written script, this code is
# 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:
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
minVal = probeVal
maxVal = probeVal
avgSum = probeVal
seenAny = True
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 there is only one item.
outData.SetDoubleQuantityData("min_val", [minVal])
outData.SetDoubleQuantityData("max_val", [maxVal])
outData.SetDoubleQuantityData("avg_val", [avgVal])

# The operation is complete.
return True
```

user-defined Outputs: Script Organization (2D and Circuit)

As described in the *Introduction* topic, the UDO scripts should all reside under the **UserDefinedOutputs** folder under either of the three library locations (system, user or personal).

Using Script Libraries

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional .NET Assemblies

Using Script Libraries (2D and Circuit)

If you decide that you need base classes or additional data files to organize your UDOs better, 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 is automatically added to the system include path (any support script files from any Lib directory through the top level UserDefinedOutputs directory can be imported) .

Using Additional .NET Assemblies (2D and Circuit)

Because the UDO functionality uses IronPython, the full .NET ecosystem is accessible. 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
   - UserDefinedDocuments\Lib,
   - UserDefinedDocuments\a\Lib,
   - UserDefinedDocuments\a\b\Lib
   - UserDefinedDocuments\a\b\c\Lib
3. Add the following line to your python script
   - Import clr
   - clr.AddReference(“com.Acme.UDOLib”)
   - import com.Acme.UDOLib –or-- from com.Acme.UDOLib import * *etc*

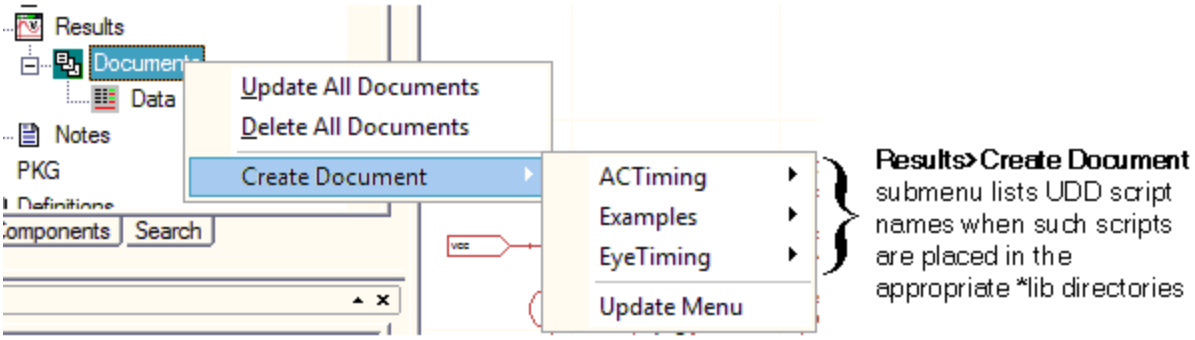
If for some reason you cannot place the .NET assemblies into a Lib directory under UserDefinedDocuments, you need to do a couple more steps before step 3 listed above.

```
Import sys

sys.path.append(“full path to your .NET assembly location”)
```

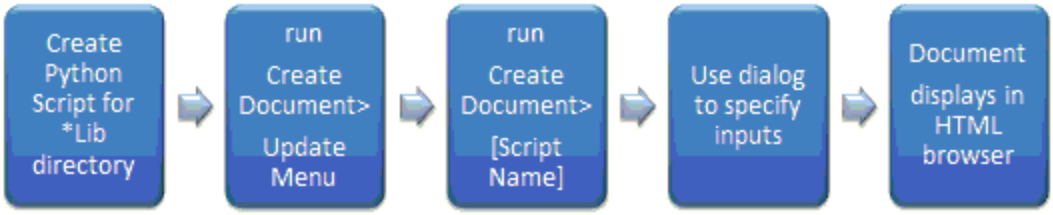

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined Documents (UDDs) (2D and Circuit)

user-defined documents (UDDs) are custom reports that you define through IronPython scripts. Once placed in a Lib directory, access the scripts via the **Create Document** command. The scripts describe a **Create User Defined Document** window that lets you specify trace and solution inputs. After you confirm your input selections, an xml, html and pdf document is generated. A web browser window opens to open the generated html file. The created document appears in the **Project Manager** window, 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.

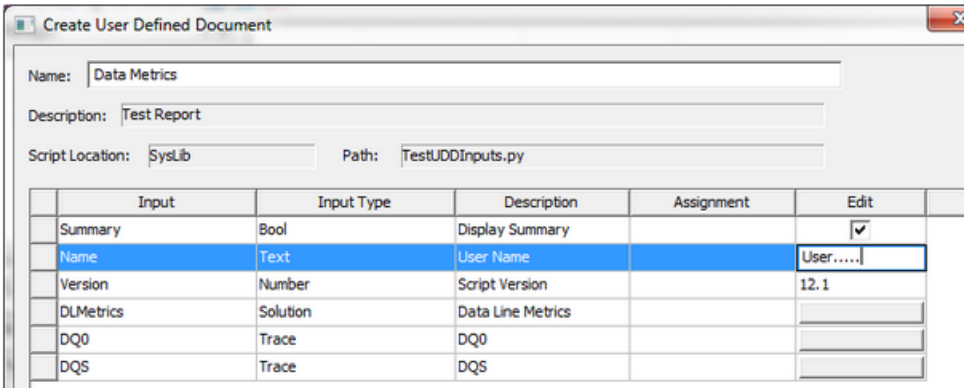
Use **Results>Create Document>Update Menu** to refresh the menu to include the new UDD scripts that have been copied to syslib, userlib or Personallib, or to exclude them if they have been deleted, after the launch of desktop.

The UDD scripts that are in syslib/UserDefinedDocuments, userlib/UserDefinedDocuments or Personallib/UserDefinedDocuments become available through the **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 window Inputs (2D and Circuit)

user-defined documents allow data from traces, solutions and report types as inputs. A UDD can specify the named inputs for which you select or enter the values in the **Create user-defined Document** window that displays when you run **Results>Create Document>** <*scriptName*>.

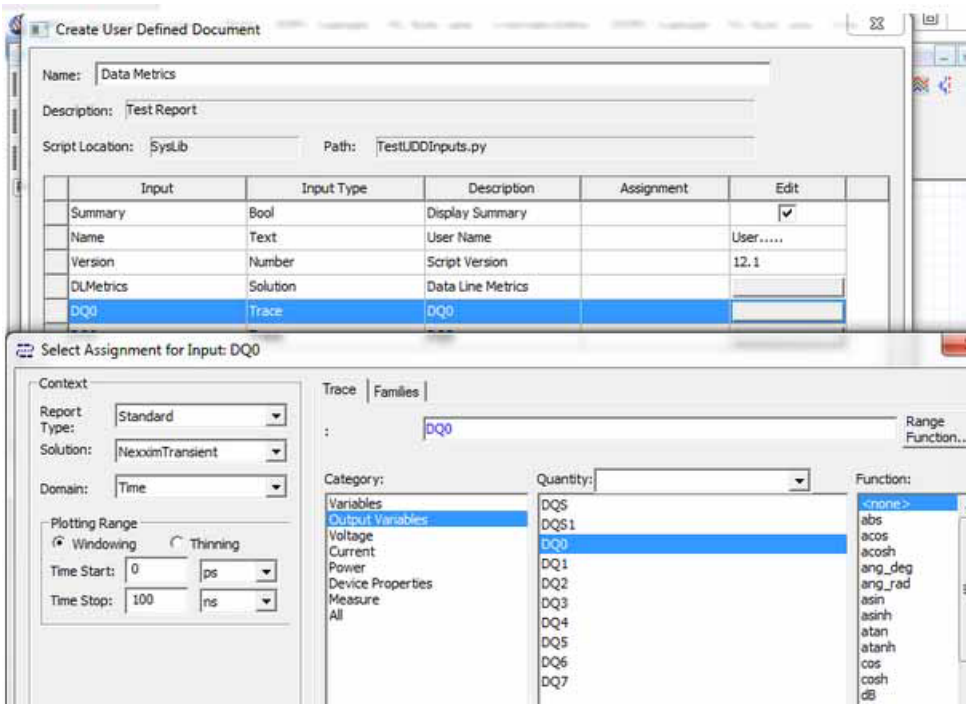


Input Types can be of Boolean, number, text, trace or solution type. The boolean, number and text type can be given a default value that can be interactively overriden when the document is created or modified. For example, select a trace when you create or modify a UDD document. The trace data is available to the user and can be accessed on the python script.

At the time of selection choose on the **Reporter** window, the report type (Standard, Eye Diagram, user-defined), solution name, context and the quantity for which you want the trace

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

data.

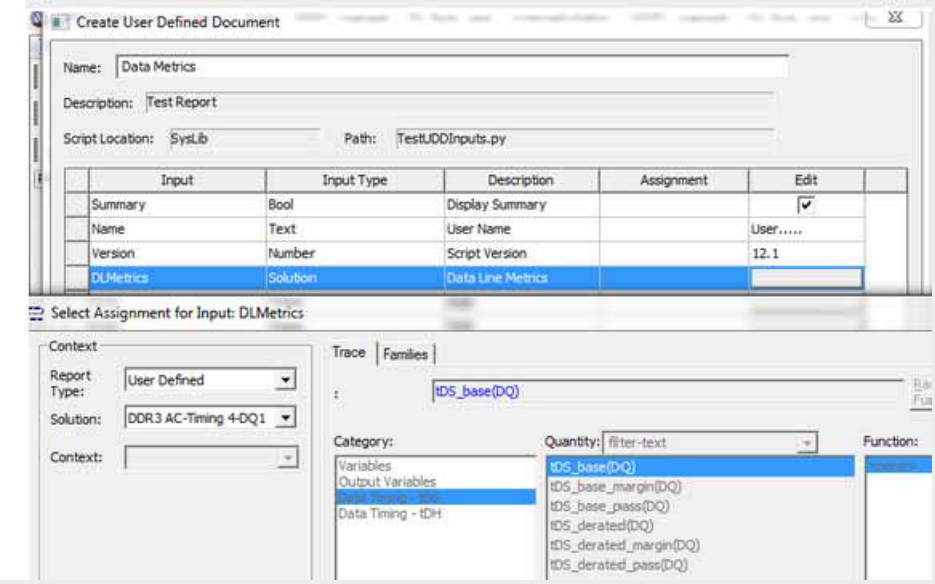


Input Type can also be Solution. You can select an entire solution when the document is created or modified. The solution data in its entirety, is now available to the user and can be accessed on the python script.

At the time of selection choose on the reporter window, 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.

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 category/Quantity/Function portion of the window is unavailable for user input.

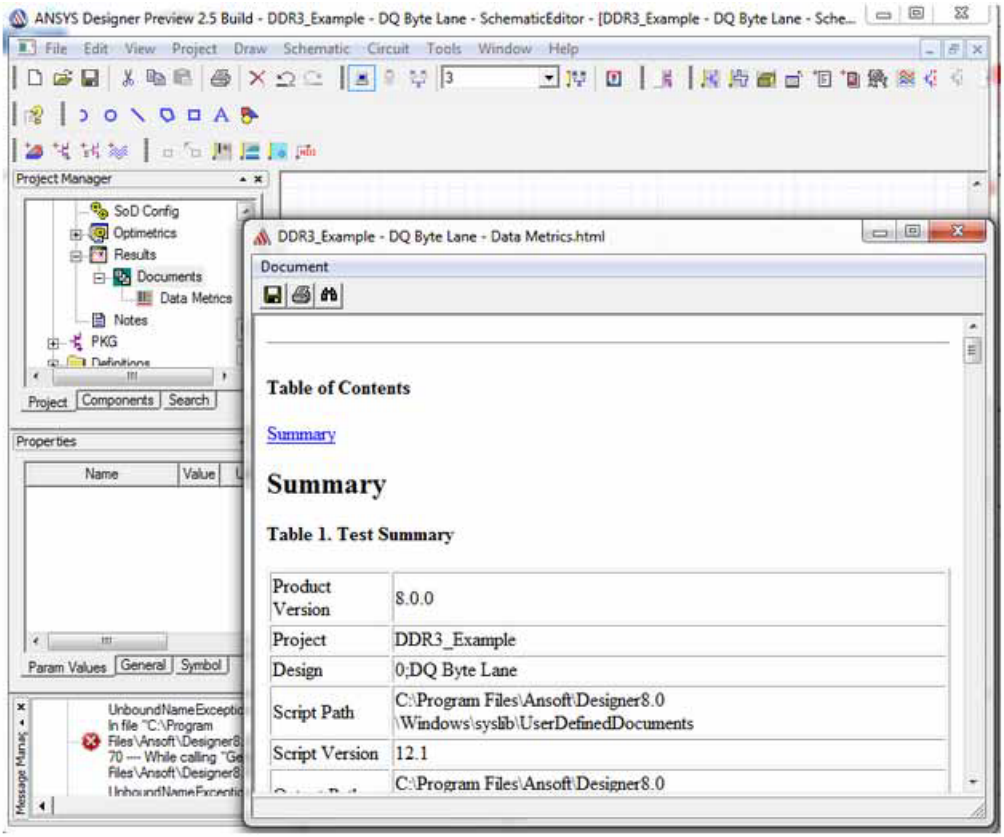


UDD Document Creation and Display (2D and Circuit)

After all the input selections for a UDD are confirmed, based on the script, an xml, html and pdf document is 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 explain in a following topic). A web browser window also opens to open the generated html file.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

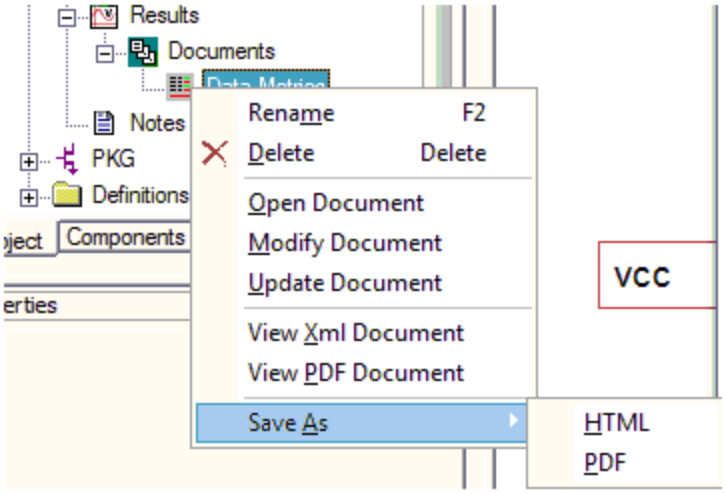
The created document is placed under a new folder named "Documents" under the "Results" folder. All documents that are created by the user for the design is placed under this folder.



Managing Documents Listed in the Project Manager window Under Results (2D and Circuit)

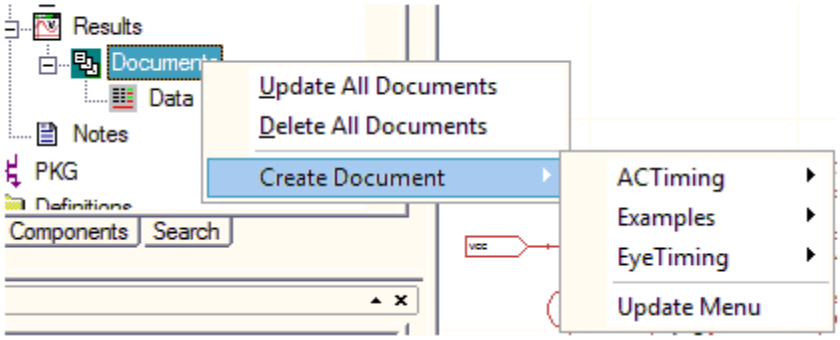
In the **Project Manager** > **Results** > **Documents** folder, right-click a user-defined document and select **Rename** and **Delete**, among other choices. **Open Document** opens the web browser with the html document. **Modify Document** opens the setup window where change the selections for the input. Click **View xml Document** or **View PDF Document** to view the document in the appropriate format, or choose **Save As** 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 shortcut menu

Right-click the Documents folder to **Update All Documents**, **Delete All Documents**, or **Create Document**.

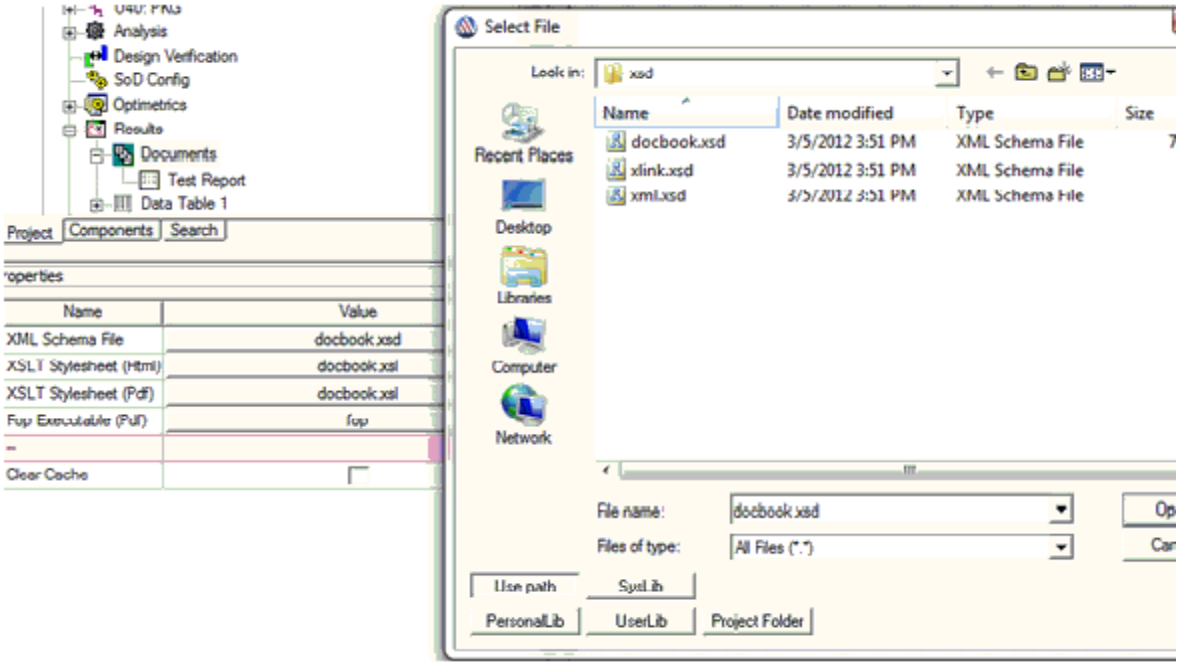


Document Folder Property window

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.



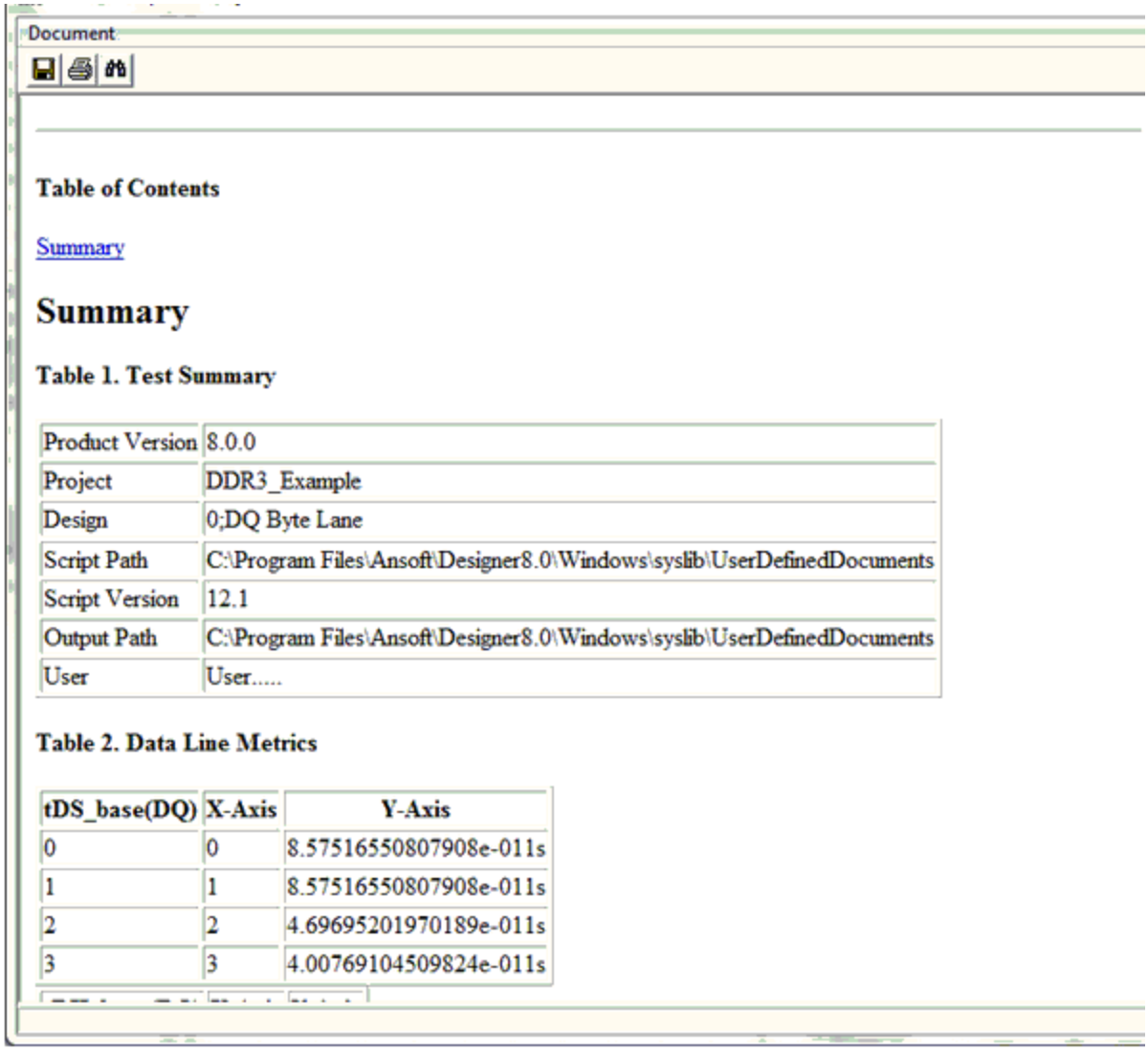
The xml, HTML and PDF generation require the xml schema file and XSLT stylesheets to generate proper output. The PDF generation also requires a FOP executable. You can use the defaults provided in the installation or provide the file paths of your own preferred stylesheets and fop executable installed in his machine.

Viewing UDDs with an HTML Web Browser (2D and Circuit)

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

document.



UDD Script Libraries (2D and Circuit)

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 is automatically added to the system include path (any support script files from any Lib directory through the top level UserDefinedDefinitions directory can be imported).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The UDD functionality uses IronPython to provide 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\UserDefinedDefinitions\a\b\c\myudd.py and have a .NET assembly called com.Acme.UDDLib You can keep the .NET assembly under
   - UserDefinedDefinitions\Lib,
   - UserDefinedDefinitions\a\Lib,
   - UserDefinedDefinitions\a\b\Lib
   - UserDefinedDefinitions\a\b\c\Lib
3. Add the following line 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 UserDefinedDefinitions, 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 Definitions: Python Script API (2D and Circuit)

A user-defined Definition (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.

Import Statements (2D and Circuit)

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.Ansys.DocGeneratorPluginDotNet.DocGenerator.API.Data
import *
from Ansys.Ansys.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.

UDDExtension Class (2D and Circuit)

The UDD itself should be implemented as an IronPython class called **UDDExtension** which must derive on the **IUDDPluginExtension** abstract base class (on the **Ansys.Ansys.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 on 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:**
>
> All of the above text has been copied on the Help topic for the UDOs and modified for the UDDs. Since the UDDs are modelled after the UDOs, the usage is also similar.

IUDDPluginExtension Abstract Class (2D and Circuit)

Required functions:

The **IUDDPluginExtension** abstract class declares the following abstract methods that must be implemented in the UDDExtension class or one of its base classes. Not implementing any of these methods results in a run-time error and a non functioning UDD.

GetUDDName() : Return a string that is used as a prefix for all solution instances created using this UDD.

Example:

```
def GetUDDName(self):
return "MinMaxAvg"
```

GetUDDDescription() : Returns a description for the UDD, its purpose etc.

Example:

```
def GetUDDDescription(self):
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
return "Sample UDD”
```

ShowDefaultSetupwindow() : Returns True if the default window is to be shown. Return False if the user does not want the default window. In this case the user might want to implement/show a customized setup window.

Example:

```
def ShowDefaultSetupwindow(self):
return True
```

GetUDDInputParams(IUDDInputHelper inputHelper, List<UDDInputParams> uddInputs) : Returns the list of inputs parameters for the User Defined Document. Returns boolean: True on success, False on failure.

The supplied input parameters are used to populate details of the parameters to which the UDD user specifies value, specify the input names and their types.

inputHelper: This is the input helper object provides methods to gather information to define the inputs. This interface is explained in the Document Generator Interface document.

uddInputs: .NET list of UDDInputParams objects. The UDD script is expected to add one instance of UDDInputParams for each input definition it wants displayed. The UDD user can, when creating the UDD, assign a matching value to each such input.

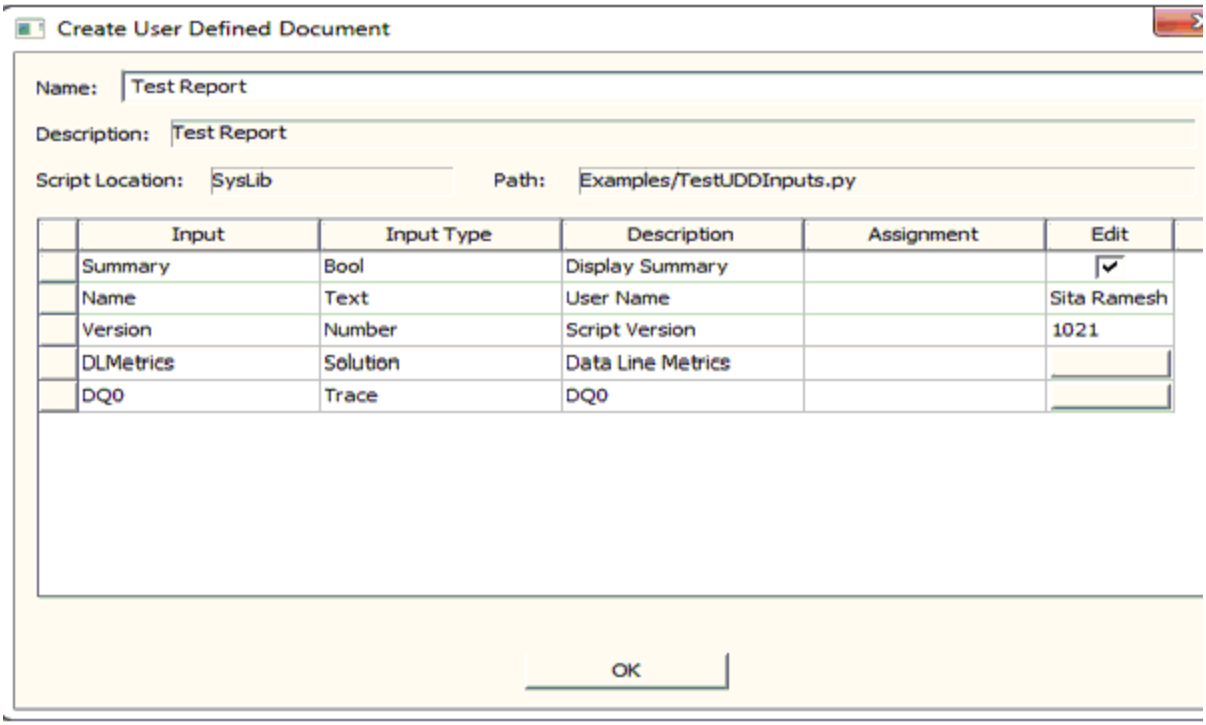
Example:

```
def GetUDDInputParams(self, inputHelper, uddInputs)
# Boolean input
param1 = UDDInputParams("Summary","Display Summary",
Constants.kBoolTypeStr, True)
uddInputs.Add(param1)
# Text input
param2 = UDDInputParams("Name","User Name", Constants.kTextTypeStr ,
"Sita Koresh")
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)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Trace input
param5 = UDDInputParams("DQ0","DQ0",Constants.kTraceTypeStr)
uddInputs.Add(param5)

return True
```

Based on the input params the following window is displayed when you click **Reports>Create Document**. The name and description of the UDD are also displayed in this window.



Generate(List<UDDInputData> uddInputs, IUDDGenerator generator, IProgressMonitor progressMonitor) : This is the main method which accesses the data on the uddInputs and generates the document.

uddInputs: The list of inputs that the user setup in the window. They are now available to query for data.

generator: This is the document generator object used to create different elements of the document like titles, topics, tables, images and write the data too. This interface is explained in the Document Generator Interface document.

progressMonitor : IProgressMonitor object. This can be used to set progress for long running calculations, check for user initiated abort etc.

Example:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
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)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 oDesign = self.GetUDDDesignContext()
 if oDesign != None:
 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
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional functions:

SetupUDDInputParams(IUDDInputHelper inputHelper, List<UDDInputParams> uddInputs) : Displays a customized window and returns the user choices for the input params.

inputHelper: This is the input helper object provides methods to gather information to define the inputs. This interface is explained in the Document Generator Interface document.

uddInputs– .NET list of UDDInputParams objects with values for each of them. These can be the user choice for each input obtained through a custom window or some other non graphical assignment.

There is no way of assigning solution data to the input without the invocation of the reporter window, so trace and solution types of input with a custom window cannot be processed.

Example:

```
def SetupUDDInputParams(self, inputHelper, uddInputs)
uddwindow = BaseExampleUDDwindow()
if uddwindow.Showwindow() == Forms.windowResult.OK:
  # Boolean input
  param1 = uddwindow.GetInput("Summary")
  uddInputs.Add(param1)

  # Text input
  param2 = uddwindow.GetInput("Name")
  uddInputs.Add(param2)

  # Number input
  param3 = uddwindow.GetInput("Version")
  uddInputs.Add(param3)
```

HandleUDDEvents(List<string> eventTags) : The tags associated with the event is received

by plugin using this abstract class.

This method is the event handler for all link events set by the SetEventLink() method on a IUDDText. (See the definition of the IUDDText object in the Document Generator Interface document).

Example:

```
def HandleUDDEvents(self, uddLinks):
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
if uddLinks[0] == "Open" Report":
   # Get Design Name
   oDesign = self.GetUDDDesignContext()
    if oDesign != None:
      oDesign.OpenReport(uddLinks[1])
 return True
```

GetUDDSchema() : Returns the file path of the schema to validate the xml. This overrides the default schema used. Return string containing the full file path of the schema.

```
def GetUDDSchema(self):
return "C:\\Program Files\\ANSYS
Inc\\v261\\AnsysEM\\common\\docbook\\schema\\xsd\\docbook.xsd"
```

GetUDDStyleSheetForHtml() : Returns the file path of the style sheet used to generate the html document. This overrides the default stylesheet for html. Returns string containing the full file path of the style sheet.

```
def GetUDDStyleSheetForHtml(self):
return "C:\\Program Files\\ANSYS
Inc\\v261\\AnsysEM\\common\\docbook\\"
```

GetUDDStyleSheetForPdf() : Returns the file path of the style sheet used to generate the pdf document. This overrides the default stylesheet for pdf. Returns string containing the full file path of the style sheet.

Example:

```
def GetUDDStyleSheetForPdf(self):
return "C:\\Program Files\\ANSYS
Inc\\v261\\AnsysEM\\common\\docbook\\xsl\\fo\\docbook.xsl"
```

GetFopExecutable() : Returns the file path of the fop executable used to generate the pdf document. This overrides the default stylesheet for pdf.Returns string containing the full file path of the fop executable.

Example:

```
def GetFopExecutable(self):
return "C:\\Program Files\\ANSYS
Inc\\v261\\AnsysEM\\common\\ApacheFOP\\fop-1.0\\fop"
```

GetUDDAppContext() : Returns the UDD Owner (if set). This is a Dispatch wrapper that is essentially a COM IDispatch implementation and corresponds to the IDispatch pointing to the **Electronics Desktop** (**Electronics Desktop**) app.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetUDDDesignContext() : Returns the UDD Owner (if set). This is a Dispatch wrapper that is essentially a COM IDispatch implementation and corresponds to the IDispatch pointing to the Design.

Data Types Used in Python Script (2D and Circuit)

There are several types that you must use while authoring the python script. Some of them are used to pass data from UI to python script and provide interface for working with this data. Some are used to pass data from python script to UI.

To pass data from python script to UI the objects of the C# class must be created in python script using their C# constructors. The objects can then be set as functions return values or set to the output parameters using their API.

Constants Class (2D and Circuit)

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

UDDInputParams Class (2D and Circuit)

The objects of this class must be created in python script in the **GetUDDInputParams()** function and the **SetUDDInputParams()** function.

Attributes :

Input Name (string)

Input Description (string)

Input Type (Can be Boolean, Number, Text, Trace or Solution) (string)

BoolData (boolean)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

IProgressMonitor Abstract Class (2D and Circuit)

The object of this class is a progress monitor. It is used to display calculations progress in UI and check is the user has requested an abort of the computation.

When displayed in the application, each progress message has four items:

A task name

A sub-task 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 progress amount

A button to abort the task in progress.

All of this functionality and abort interaction is achieved using the following functions.

SetTaskName (string taskName)

SetSubTaskName (string subTaskName)

BeginTask (string name)

SetTaskProgressPercentage(int progressPercent)

CheckForAbort(): If the quantities being generated are computationally expensive, the UDO author can periodically call this method , then call EndTask with Fail and return False.

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)
```

UDD Input Interfaces (2D and Circuit)

The Generate function takes in a list of inputs. These input interfaces allow the user to access data on the design.

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 window.

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 window.

IUDDInputText : This interface exposes 3 methods

Name() : Gets the inputs name.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type() : Gets the input type.

Data() : Gets the text data, set by the user in the setup window.

IUDDInputTrace: This interface exposes 3 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 3 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 dataset by the user
boolinput = input[0].Data()
# Getting the double dataset by the user
dblinput = input[1].Data()
# Getting the text dataset by the user
textinput = input[2].Data()
# Getting the category names in a solution
categories = input[3].CategoryNames()
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Getting the quantity names based on a category
quantities = input[3].QuantityNames(categories[0])
# Getting the XY data on the trace
xydata = input[4].DoubleData()
```

user-defined Document Scripting Interface (2D and Circuit)

To access the UserDefineddocuments scripting object, use:

```
Set oModule = oDesign.GetModule("UserDefinedDocuments")
```

Once you have the scripting object, use the following methods:-

1. AddDocument ([in] VARIANT data, [in] VARIANT traces, [out, retval] BSTR* uniqueName)
    a. Takes a VARIANT data which defines the document.
    b. Takes a VARIANT trace data for the inputs in the document.
    c. Returns a unique name
2. EditDocument([in] BSTR originalName, [in] VARIANT modifiedData, [in] VARIANT modifiedTraces, [out, retval] BSTR* uniqueName)
    a. Takes the name of the original document.
    b. Takes a VARIANT data which defines the edited document.
    c. Takes a VARIANT trace data for the inputs in the document.
    d. Returns a unique name
3. RenameDocument([in] BSTR oldName, [in] BSTR newName)
    a. Takes the name of the original document.
    b. Takes the new name of the document.
4. DeleteDocument([in] BSTR name)
    a. Takes the name of the document to be deleted.
5. UpdateDocument([in] BSTR name)
    a. Takes the name of the document to be updated.
6. ViewHtmlDocument([in] BSTR name)
    a. Takes the name of the document to be viewed in HTML.
7. ViewPdfDocument([in] BSTR name)
    b. Takes the name of the document to be viewed as a PDF.
8. SaveHtmlDocumentAs([in] BSTR name, [in] BSTR saveTo)
    a. Takes the name of the document to be saved.
    b. Takes the file path to save the document as.
9. SavePdfDocumentAs([in] BSTR name, [in] BSTR saveTo)
    a. Takes the name of the document to be saved.
    b. Takes the file path to save the socument as.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. GetDocumentNames([in] BSTR separator, [out, retval] VARIANT* names)
    a. 'separator' is used to convey the directory "level"
    b. Returns the names of documents.
11. GetDocumentDefinitionNames([in] BSTR separator, [out, retval] VARIANT* names)
    a. 'separator' is used to convey the directory "level"
    b. Returns the (file) names of doc definitions according to the files in various installation directories.
12. DeleteAllDocuments()
13. UpdateAllDocuments()

For 6, 7, 8, and 9, the document must have an existing, generated HTML or PDF.

The UserDefinedDocument Data Format (2D and Circuit)

To define a document in Python script:

```
 [
"NAME:Test Report", "Test Report", "SysLib",
"Examples/TestUDDInputs",
[
"NAME:Inputs",
[
"NAME:DLMetrics","Solution", "Data Line Metrics", -1, -1
],
[
"NAME:DQ0", "Trace", "DQ0", -1, -1
],
[
"NAME:DQS", "Trace", "DQS", -1, -1
],
[
"NAME:Name", "Text", "User Name", ["Sita Ramesh"]
],
[
"NAME:Summary", "Bool", "Display Summary", [True]
],
[
"NAME:Version", "Number", "Script Version" [1021]
]
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
]
 ],
 [
"NAME:DocTraces",
[
"NAME:DLMetrics",
[
"user-defined", "", "DDR3 AC-Timing 4-DQ1",
[
"Context:=" , ""
],
[
"Index:=" , ["All"], "Trise:=" , ["Nominal"], "Tfall:=" ,
["Nominal"], "Pulse_Width:=" ,
["Nominal"], "Data_Rate:=" , ["Nominal"], "Length:=" , ["Nominal"]
],
[
"Probe Component:=" , [""]
],
[]
]
],
[
"NAME:DQ0",
[
"Standard", "DQ0", "NexximTransient",
[
"NAME:Context", "SimValueContext:=" ,
[1,0,2,0,False,False,-
1,1,0,1,1,"",0,0,"DE",False,"0","DP",False,"20000000","DT",False,"0.0
01","WE",False,"100ns","WM",False,"100ns","WN",False,"0ps","WS",False
,"0ps"]
],
[
"Time:=" , ["All"],"Trise:=" , ["Nominal"],"Tfall:=" ,
["Nominal"],"Pulse_Width:=" ,["Nominal"],
"Data_Rate:=" , ["Nominal"], "Length:=" , ["Nominal"]
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
],
[
"Probe Component:=" , ["DQ0"]
],
[]
]
],
 ]
```

Document Generator Interfaces (2D and Circuit)

This document briefly describes the API interfaces available in the document generator plugin.

(Ansys.Ansys.DocGeneratorPluginDotNet.dll)

Scripting objects available in the script for the Generate function

- oApp = self.GetUDDAppContext(

  Gets the application context

  Usage:- Gets 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

  Usage:- Gets 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set the document output file path.

```
docgen.SetOutput("C:\\Examples\\DocumentOutput.xml")
```

2. Get the document root.

```
docroot = docgen.GetDocumentRoot()
```

3. Write out to the output file.

```
docgen.Write()
```

4. Write Html document

```
void WriteHTML();
```

5. Write PDF document

```
void WritePDF();
```

6. Load the Html transform object

```
void LoadHTMLTransform();
```

7. Load the cached PDF transform object

```
void LoadPDFTransform();
```

8. Get script path

```
string GetScriptPath();
```

9. Get output file path

```
string GetOutputFilePath();
```

- IUDDRoot interface

Calling GetDocumentRoot() on the IUDDGenerator interface provides you with the this interface. This interface can be used to

1. Add a new topic to the document. Provide a title.

```
section1 = docroot.AddSection("Section title")
```

2. Add a new topic to the document. Provide a title and subtitle

```
section1 = docroot.AddSection("Section title", "Section subtitle")
```

3. Add a new title

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
section1 = docroot.AddTitle("Title")
```

4. Add a new subtitle

```
section1 = docroot.AddSubtitle("Subtitle”)
```

- IUDDSection interface

Calling AddSection() on the IUDDRoot interface provides you with the this interface. This interface can be used to

1. Set an ID for the topic 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 the this interface. On this interface call the following methods

1. Set an ID for the image for internal links.

```
image1.SetID("id")
```

2. Set alignment information can be "center", "left" and "right".

```
image1.SetAligment("center")
```

3. Set the file path of the image file. Not necessary if image file path is set through the AddImage() method

```
image1.SetFileRef("Image path")
```

4. Set the format of the image file. Can be "BMP", "PNG", "JPEG", "JPG", "DVI", etc.

```
image1.SetFormat("format")
```

- IUDDText interface

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calling AddText() on the IUDDSection interface provides you with the this interface. On this interface 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. On this interface call the following methods

1. Set an ID for the table for internal links.

```
table1.SetID("id")
```

2. Set alignment can be "center", "left" and "right".

```
table1.SetAlignment("center")
```

3. Set the background color of the table

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
table1.SetBgColor(string bgcolor)
```

4. Set the frame type. Can be "all", "bottom", "top,", "sides", "topbot"

```
table1.SetFrame(string frame)
```

5. Add a table group and specify the number of columns. A table can have multiple table groups.

```
IUDDTableGroup table1.SetTableGroup(int columns)
```

- IUDDTableGroup interface

Calling AddTableGroup() on the IUDDTable interface provides you with the this interface. On this interface 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. Takes number of sub rows.

```
IUDDTableRow tgroup1.AddHeader(int rows)
```

6. Add a row of content to the table group

```
IUDDTableRow tgroup1.AddContent()
```

7. Add content with multiple rows to the table group. Takes number of sub rows.

```
IUDDTableRow tgroup1.AddContent(int rows)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- IUDDTableRow interface

Calling AddHeader() & AddContent() on the IUDDTableGroup interface provides you with the this interface. On this interface call the following methods

1. Set an ID for the table row for internal links.

```
trow1.SetID("id")
```

2. Set alignment information; can be "center", "left" and "right".

```
trow1.SetAlignment("center")
```

3. Set cell text. Can be cell content or header text. Takes a column index and a text string. It is added to the first row.

```
IUDDTextElement trow1.Add(int column, string text)
```

4. Set cell text. Can be cell content or header text. Takes a column index, row index and a text string. Takes in a row number because a table row can have multiple sub rows.

```
IUDDTextElement trow1.Add(int column, int subrow, string text)
```

5. Set cell content. Takes a column index and an int value. It is added to the first row.

```
IUDDTextElement trow1.Add(int column, int value)
```

6. Set cell content. Takes a column index, row index and a int value.

```
IUDDTextElement trow1.Add(int column, int subrow, string text)
```

7. Set cell text. Takes a column index and a double value. It is added to the first row.

```
IUDDTextElement trow1.Add(int column, double value)
```

8. Set cell text. Takes a column index, row index and a double value.

```
IUDDTextElement trow1.Add(int column, int subrow, double value)
```

9. Set cell text spanning 2 columns. Can be cell content or header text. Takes a sub row index , starting column index., ending column index and a text string.

```
IUDDTextElement trow1.AddSpanningcolumnst(int subrow, int
columnstart, int columnend, string text)
```

10. Set cell text. Can be cell content or header text. Takes a column index, starting sub row index, ending sub row index and a text string.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
IUDDTextElement trow1.AddpanningRows(int column, int subrowstart,
int subrowend, string text)
```

Includes all the methods exposed by the IUDDText interface.

- IUDDInputHelper interface

  The GetUDDInputParams() and SetUDDInputPrams provide this interface to gather information on the UI. On this interface call the following methods

1. Returns a list of user-defined solution names.

```
IList<string> inputHelper.UDSSolutionNames()
```

2. Returns the list context names for a given user-defined solution.

```
IList<string> inputHeper.UDSContextNames(string solutionname)
```

3. Returns the file path of the script corresponding to the user-defined solution.

```
string inputHelper.UDSScriptFileName(string solutionname)
```

4. Returns the parameters defined in the user-defined solution.

```
PropList UDSParameters(string solutionName)
```

5. Returns a dictionary of probe names and their corresponding expression, for a user-defined solution.

```
IDictionary<string, string> UDSProbes(string solutionName)
```

6. Returns a dictionary of variables and their nominal values.

```
IIDictionary<string, string> UDSVariableValues(string solutionName)
```

Technical Notes for 2D and Circuit Reports

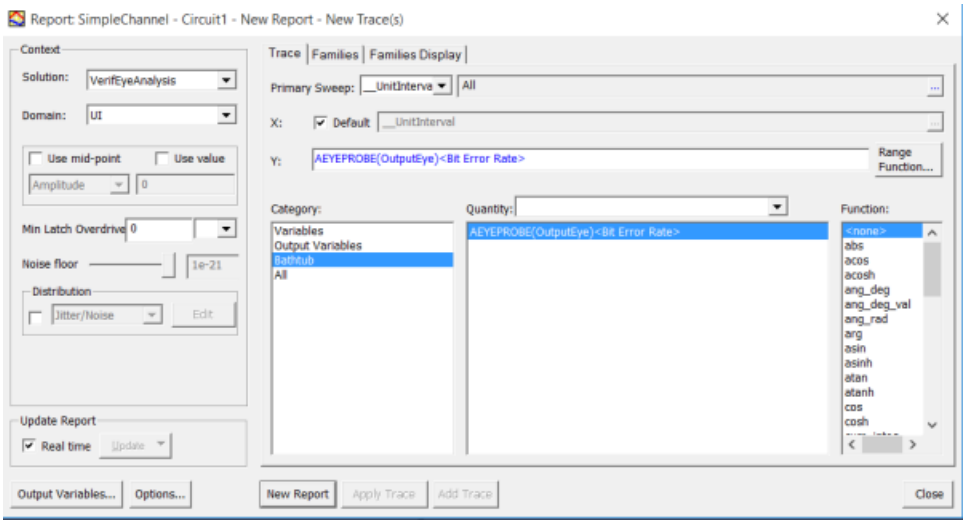
Electronics Desktop post-processing allows you to define all types of plots for simulations prior to running an analysis. Using the reporter window, define plots prior to simulation for all types of reports, including VerifEye, QuickEye, AMI, and Transient PMF. All reports are populated using the eye probes and AMI probes in a design, so set up all types of displays before an analysis.

The following topics describe how to take advantage of additional Electronics Desktop post-processing tools in order to generate various reports and view 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.

Unit Interval and Amplitude Values for Circuit Bathtub Curves

You can specify values for the Unit Interval or Amplitude while plotting bathtub curves for a QuickEye or VerifEye analysis. The controls allow you to choose between two check boxes for **Use mid-point** or **Use value**. The menu allows selection of **Amplitude** or **UI**.

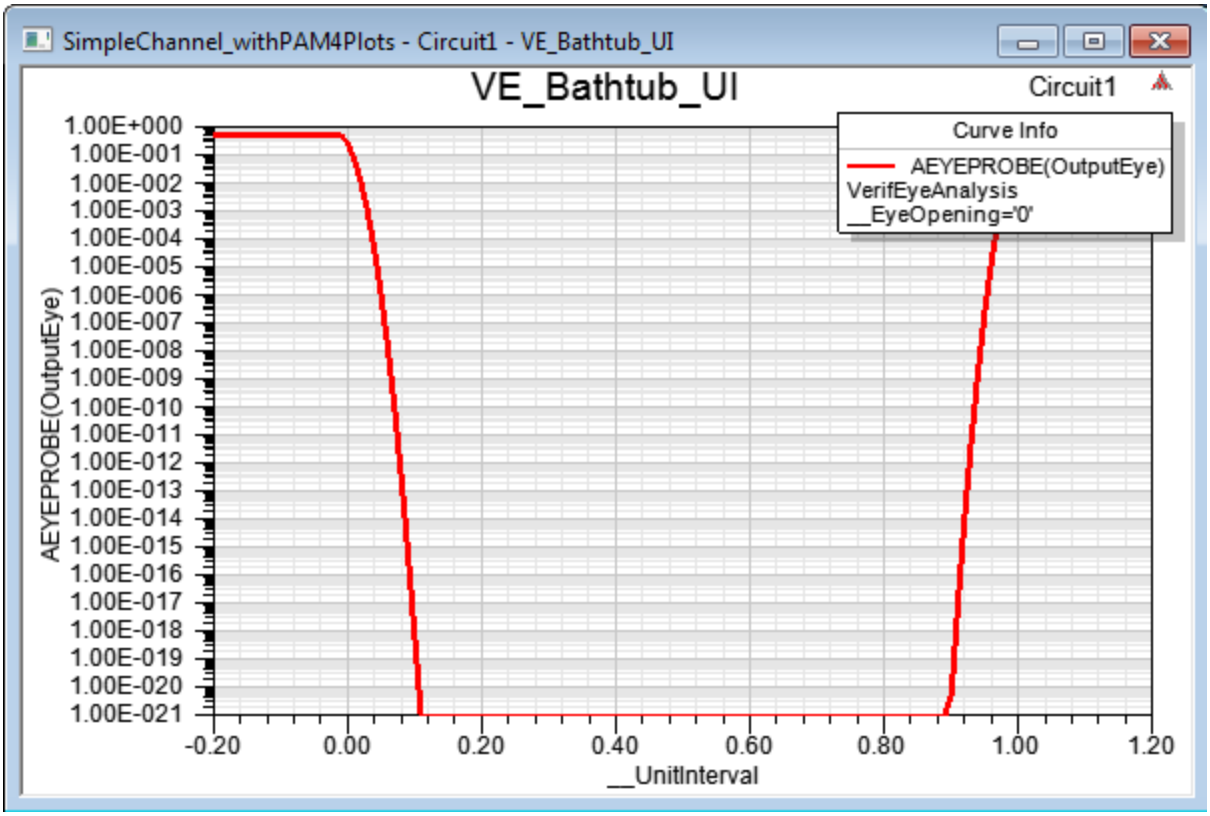


- When the **Use value** group box is checked, the edit window is enabled, allowing you to enter an Amplitude or UI value at which to see the bathtub curve. Electronics Desktop interpolates the data to plot a bathtub curve at that Amplitude or UI.
- The **Use midpoint** check box can also be enabled. Select **Amplitude** or **UI** on the menu.
- By default, the **Use midpoint** box is checked, and the midpoint of Amplitude is used as the default sweep value for the plot.
- Bathtub noise floor range is e-16 to ~ e-21.
- Default data length is set at 2^{18} or around $2.5x10^5$.
- BitPattern is PBRS and "random bit pattern generation".

> **Note:** With QuickEye, AMI, and Transient plots, when bathtub curves can be plotted and txrj is present, a warning message is displayed if the number of bits is less than 2.5e5.

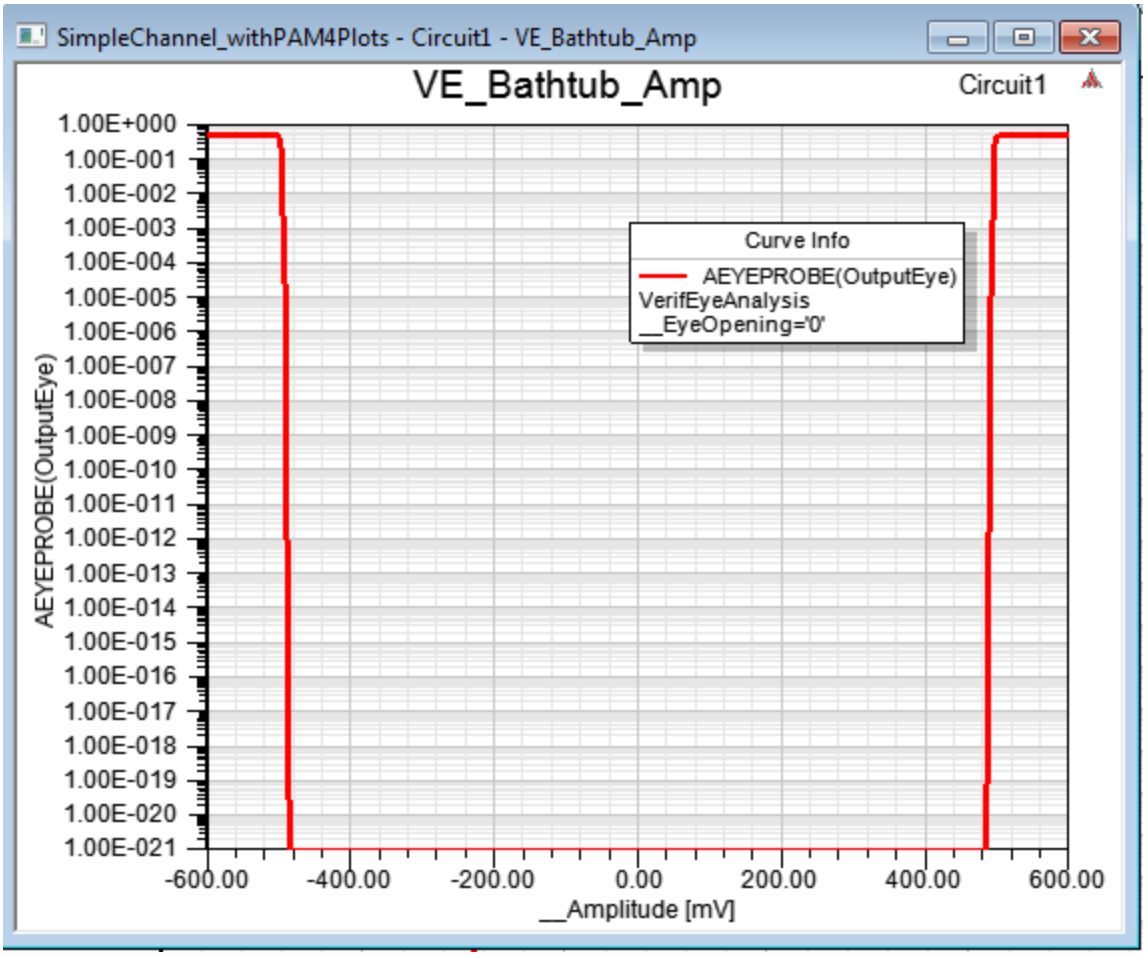
The following shows a VerifEye plot with the default settings and NRZ transmission (X-axis is the UI):

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



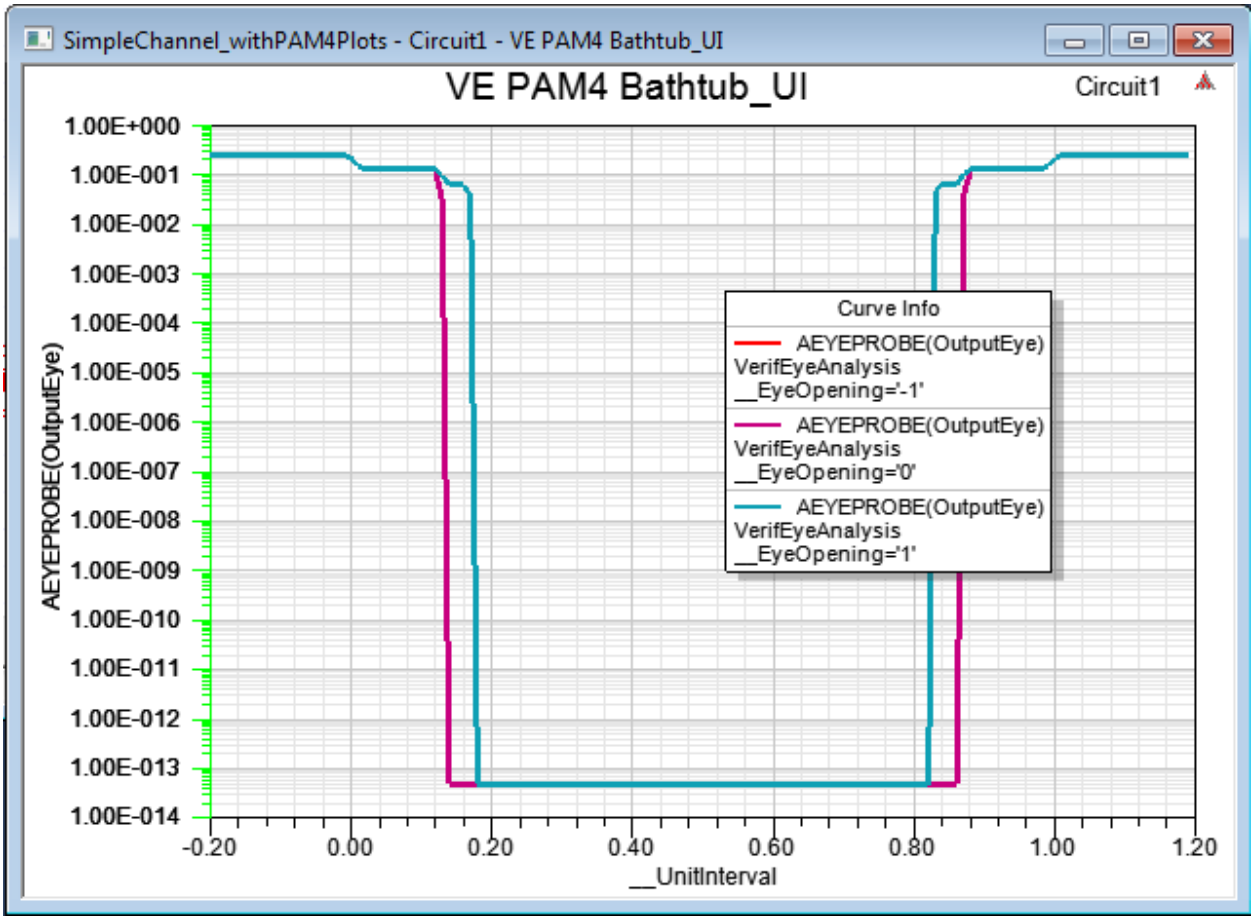
Following is a VerifEye plot centered on the UI with NRZ (X-axis is the Amplitude):

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Following is a VerifEye PAM-4 plot centered on the Amplitude (X-axis is the UI):

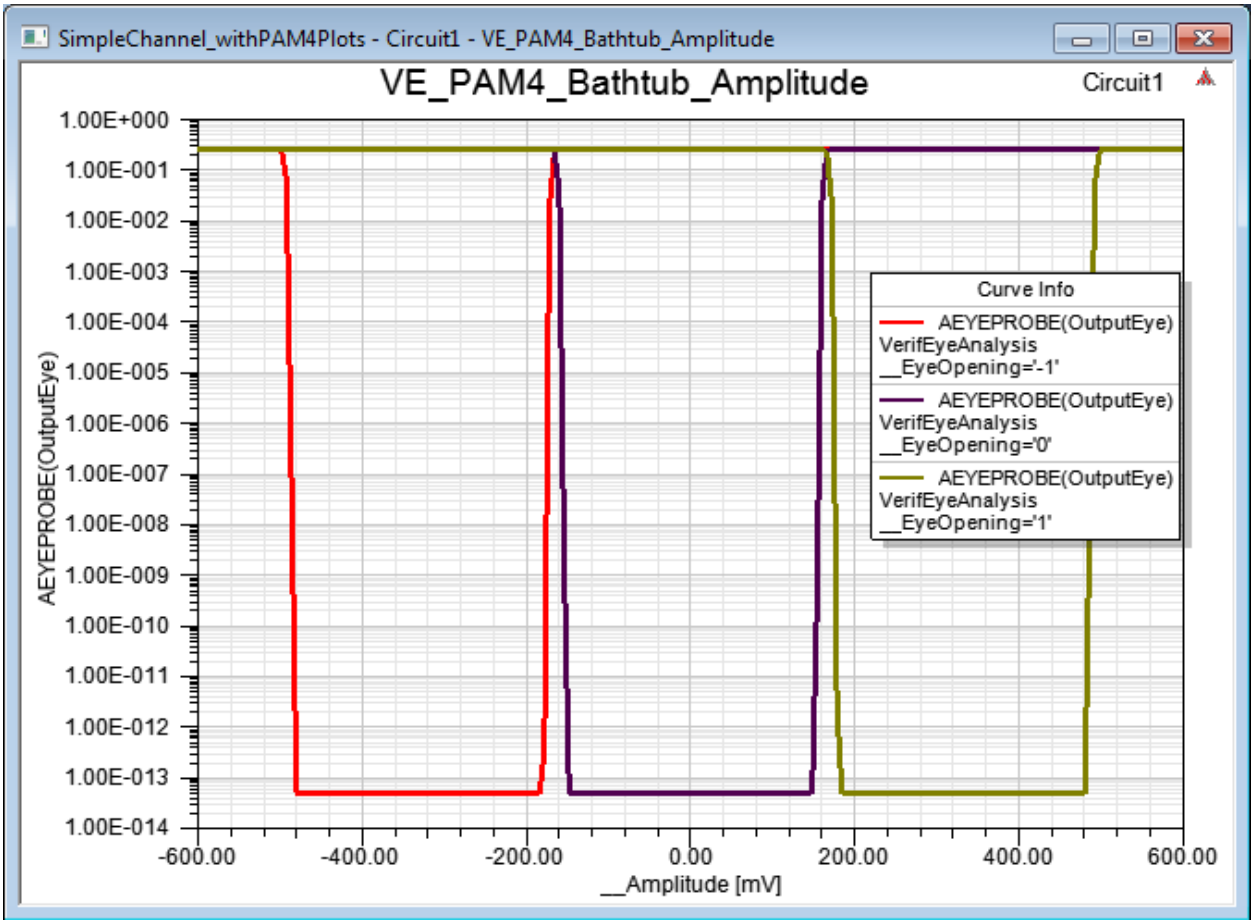
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The PAM-4 eye has three levels. The Symbol error rates for each of the three sub-eyes is shown in a different trace.

Following is a VerifEye PAM-4 plot centered on the UI (X-axis is the Amplitude):

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

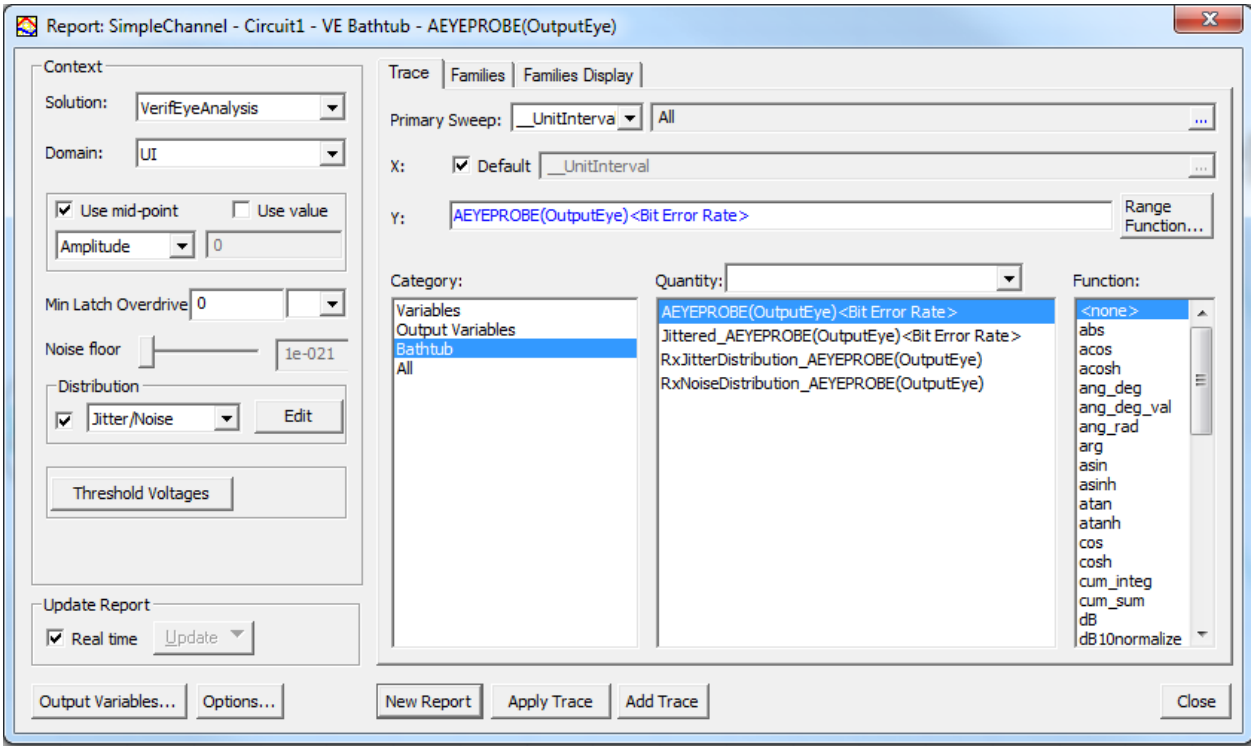


The above shows the symbol error rates for each of the three are depicted as three separate traces at different voltage levels.

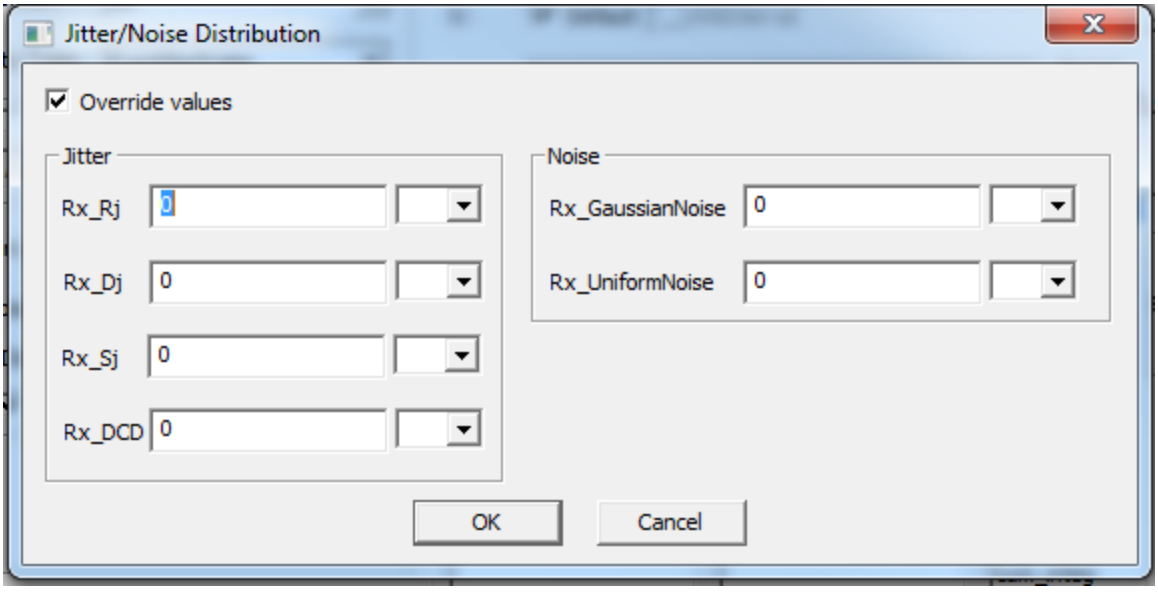
Apply Receiver Jitter and Noise Controls Simultaneously (2D and Circuit)

You can apply distributions of receiver jitter (variation in timing) and noise (variation in voltage) to the simulated data, using the controls in the **Distribution** group box of the reporter window. To view the added jitter or noise distribution, a VerifEye, QuickEye, or AMI bathtub plot is used. For AMI, the **Distribution** group box is available with the "Eye After Probe" quantity.

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 check box in the **Distribution** group box and select **Jitter/Noise** on the drop-down menu. Then click **Edit** to open the **Jitter/Noise Distribution** window.



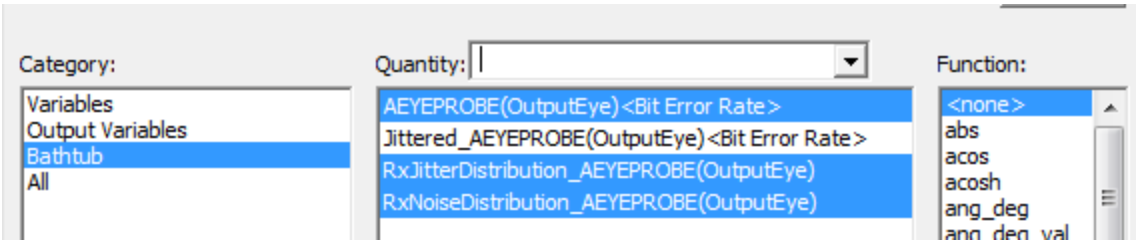
- Add a Gaussian Jitter distribution in the **Rx_Rj** field, giving the standard deviation. Specify the time units.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Add a Uniform Jitter distribution in the **Rx_Dj** field, giving the half-width of the distribution in time, and specify the time units.
- Add a sinusoidal jitter (**Rx_Sj**), giving the amplitude of the distribution in time units.
- Add a clock duty cycle distortion (**Rx_DCD**), giving one-half the peak-to-peak variation of the distortion in time units.
- Add a Gaussian Noise distribution in the **Rx_GaussianNoise** field, giving the standard deviation, and set the voltage units.
- Add a Uniform Noise distribution in the **Rx_UniformNoise** field, giving the half-width of the distribution, and specify the voltage units.
- Click **OK** to close the window.

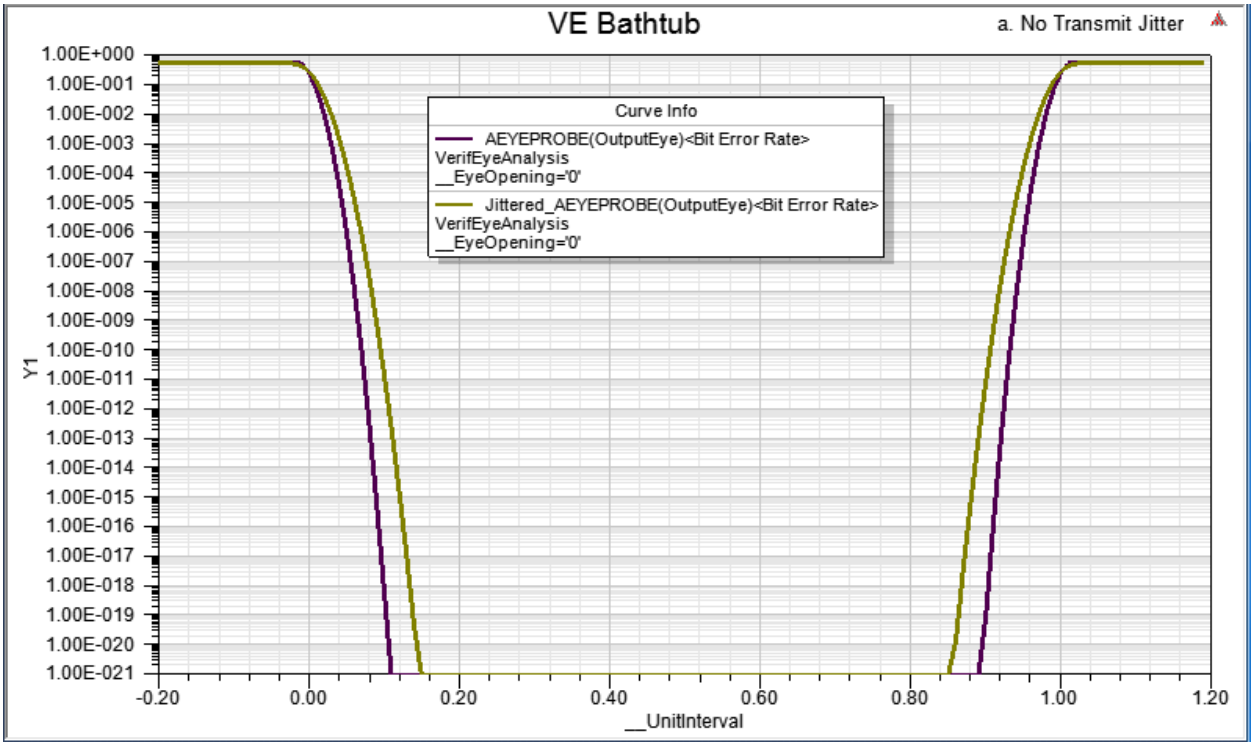
The parameter names and definitions are as defined in the IBIS specification. See AMI Receive Jitter Parameters .

When Jitter/Noise has been enabled, four quantities are available for plotting: the bathtub with no jitter (AEYEPROBE(OutputEye)<Bit Error Rate>), the bathtub with the jitter and noise applied (Jittered_AEYEPROBE(OutputEye), the distribution of the jitter (RxJitterDistribution_ AEYEPROBE(OutputEye), and the distribution of the noise (RxNoiseDistribution_AEYEPROBE (OutputEye).



The Bathtub plot shows some example results:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



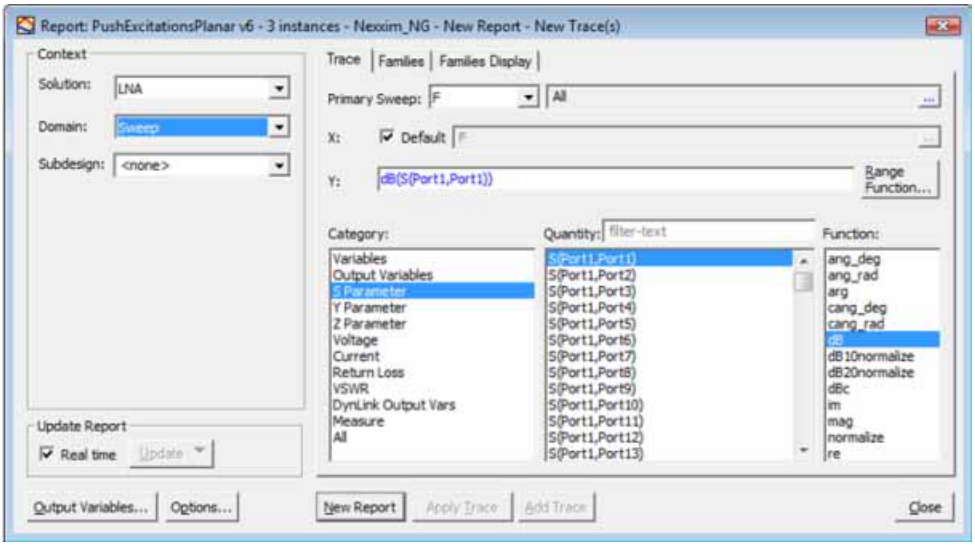
The AEYEPROBE(OutputEye) trace shows the BER curve before the jitter is applied (UI=1ns). The Jittered_AEYEPROBE(OutputEye) trace shows the BER curve after a random receive jitter (**Rx_Rj**) with standard deviation 100ps is applied.

EM Subdesign Plotting on the Top-Level Design (2D and Circuit)

For a Nexxim project with EM subdesigns, plot reports on the subdesigns directly on the top-level Nexxim design. When the reporter window is invoked on the top-level Nexxim design, it contains a drop-down menu in the **Context** group box that can specify the **Subdesign**. When the **Subdesign** selection is set to the <none> default, the domain, categories and quantities

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

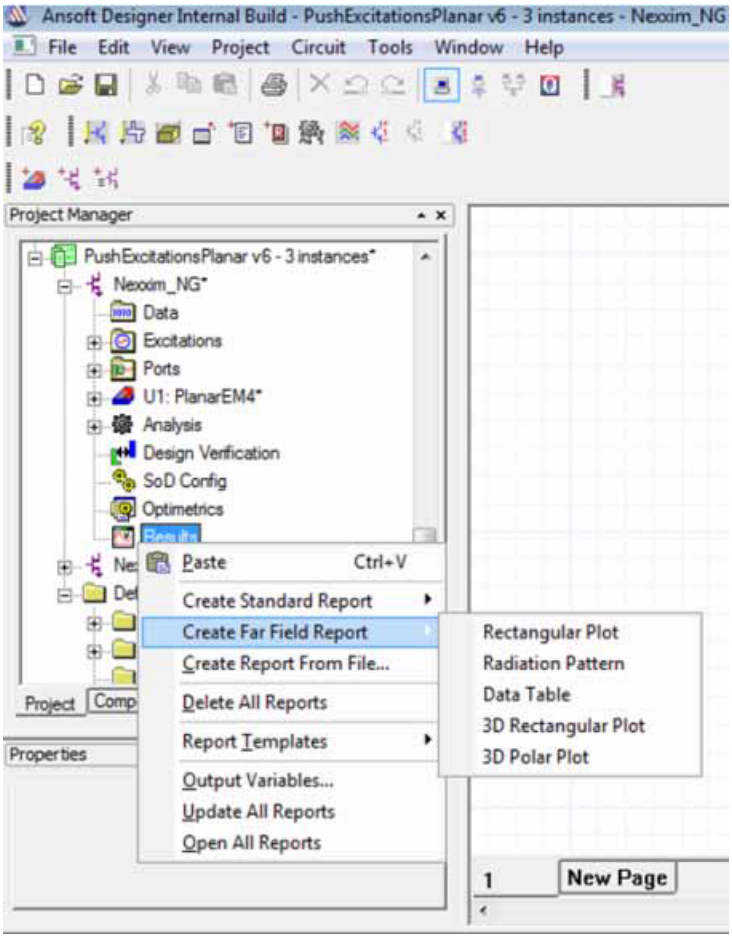
shown in the reporter window are related to the top level design.



But when the **Subdesign** selection is set, the reporter window populates with **Category** and **Quantity** values on the subdesign. This allows you to plot EM responses on the top-level reporter window, rather than editing the subdesign and opening the reporter window there.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

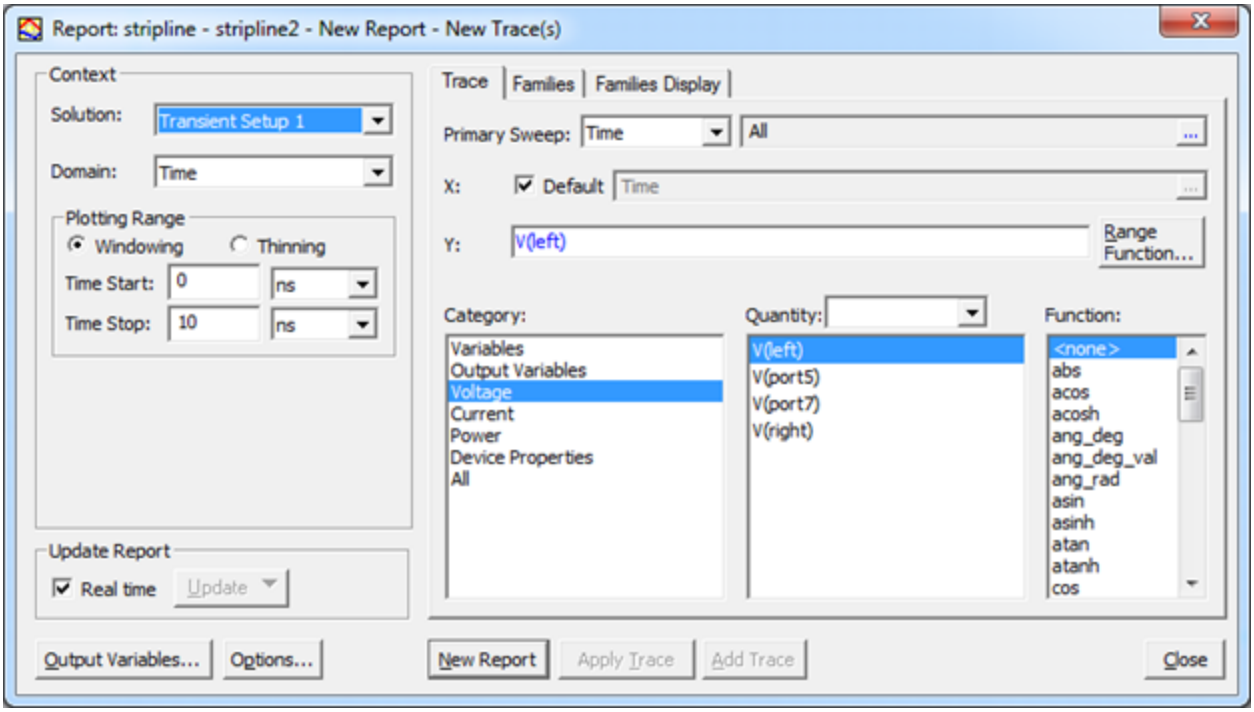
Additionally, plot far field reports from the subdesign.



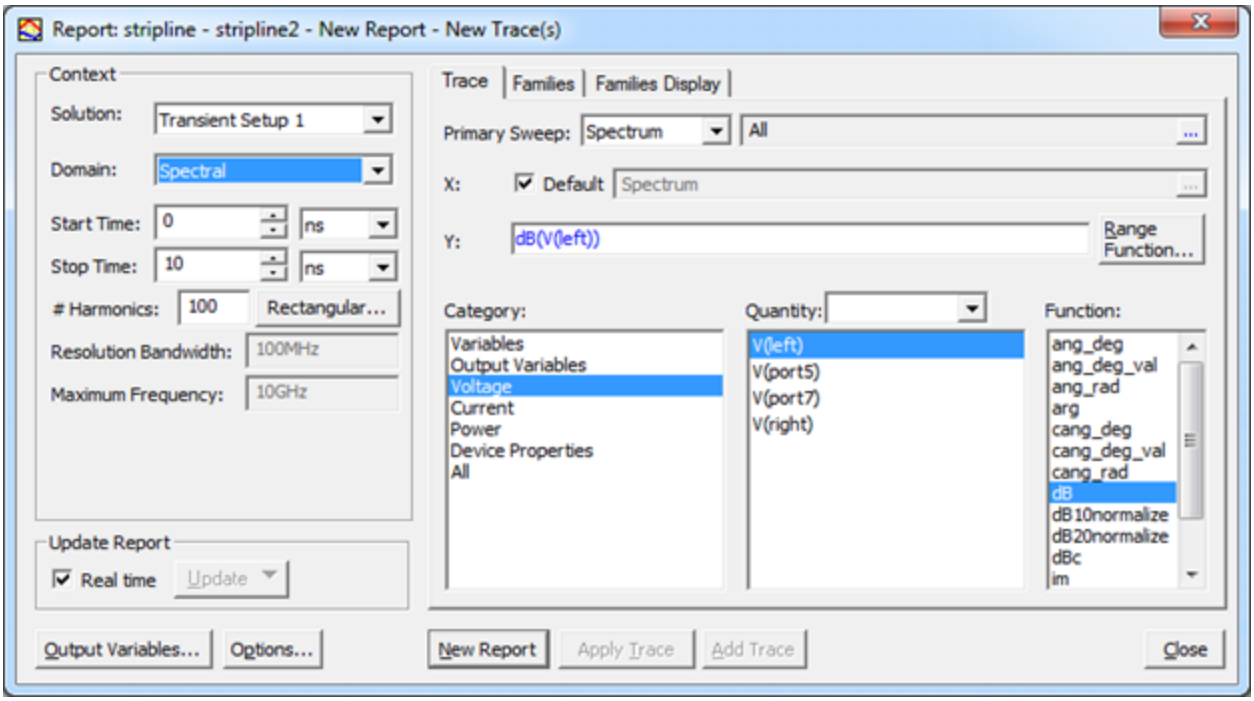
Post Processing for Transient in Layout

Transient results include voltage, currents, and power quantities in the Time domain and the Spectral domain. In the Time domain, the user can set a range to plot a window of the data or thin the data displayed in 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.

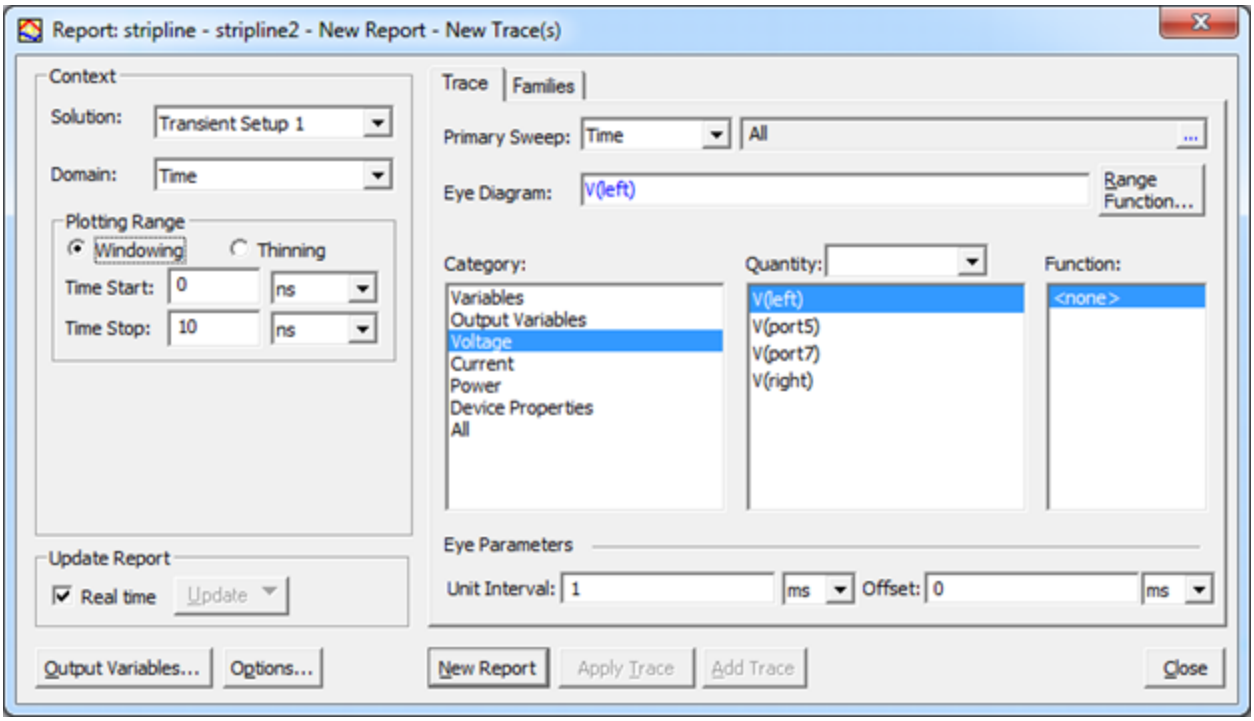


In the spectral domain, set the Start/Stop time and the number of harmonics, then plot spectral plots using the voltage, time, and power quantities.

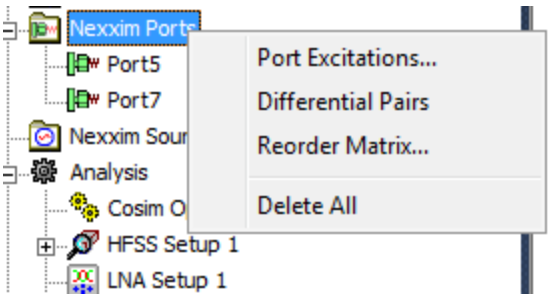


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also plot eye diagrams in Time domain.



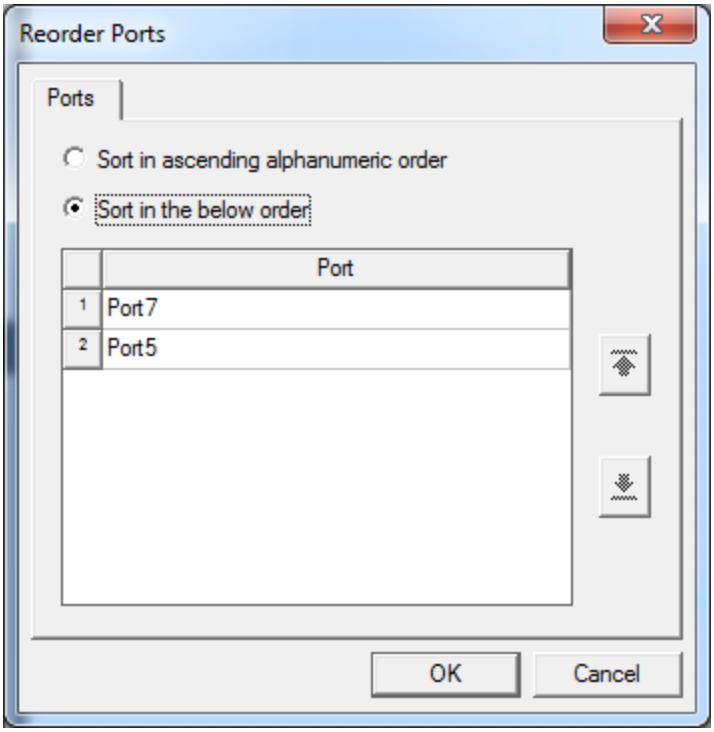
You can also use the post-processing window to reorder ports, set differential pairs, and port excitations for Interface Ports.



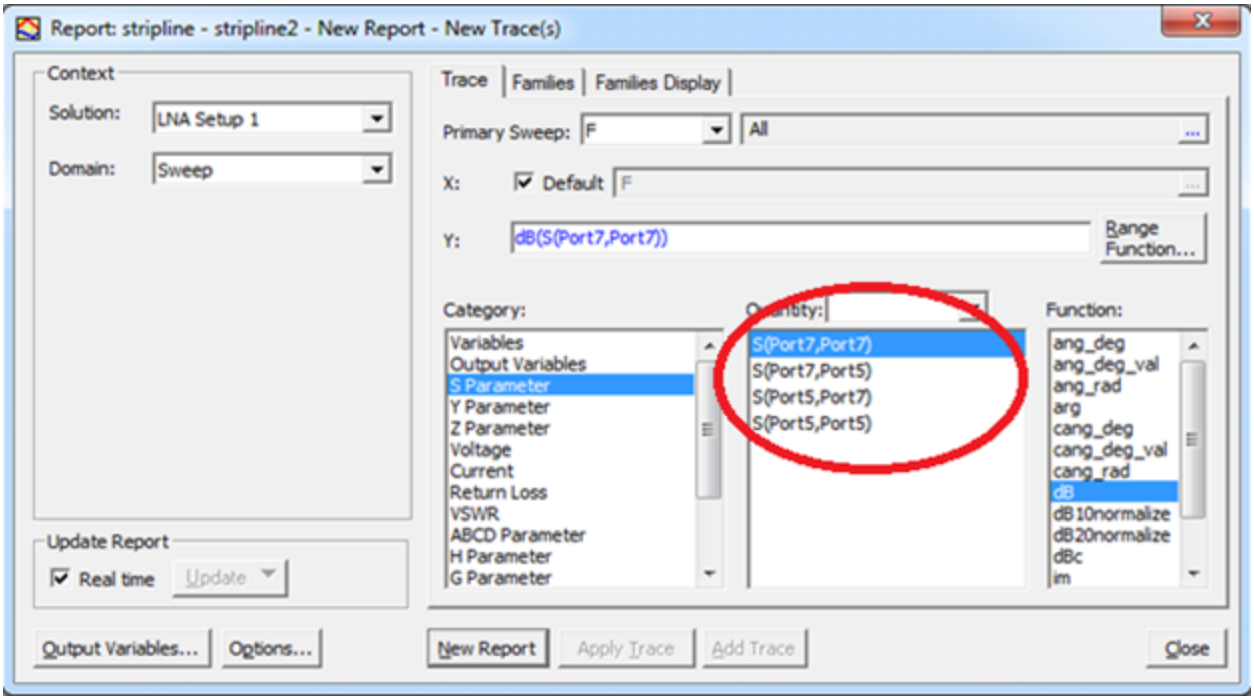
Reorder Nexxim Interface Ports

The user choose to reorder the ports in alphanumeric order or in any custom order of their choice. This order is used to reorder the results matrix and is displayed on the matrix data page, by the Network Data Explorer, and in the Reporter window.

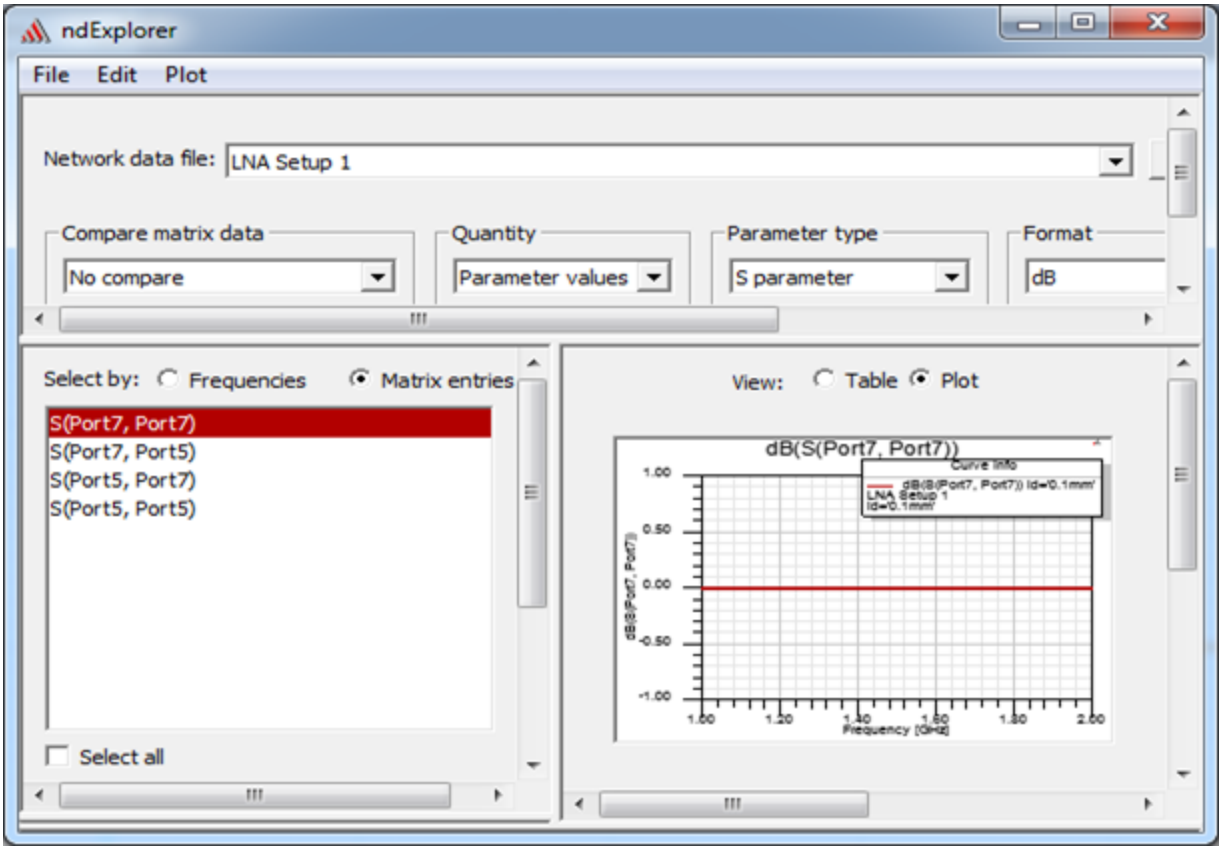
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here is the order as it is displayed in the Reporter and the Network Data Explorer.



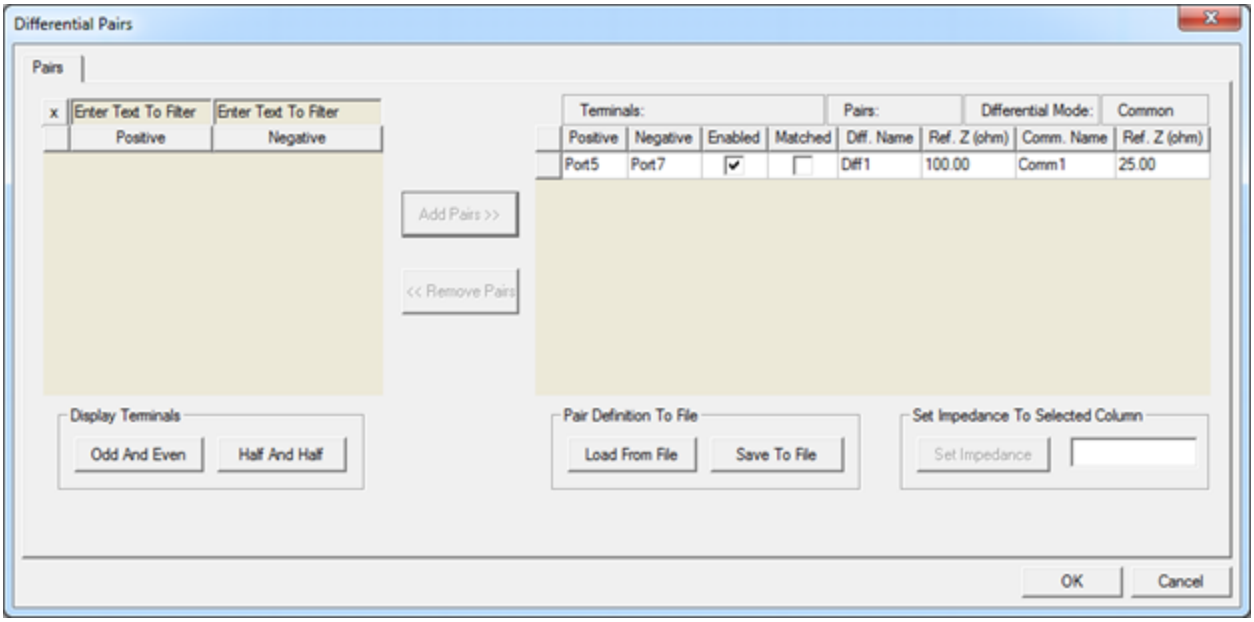
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



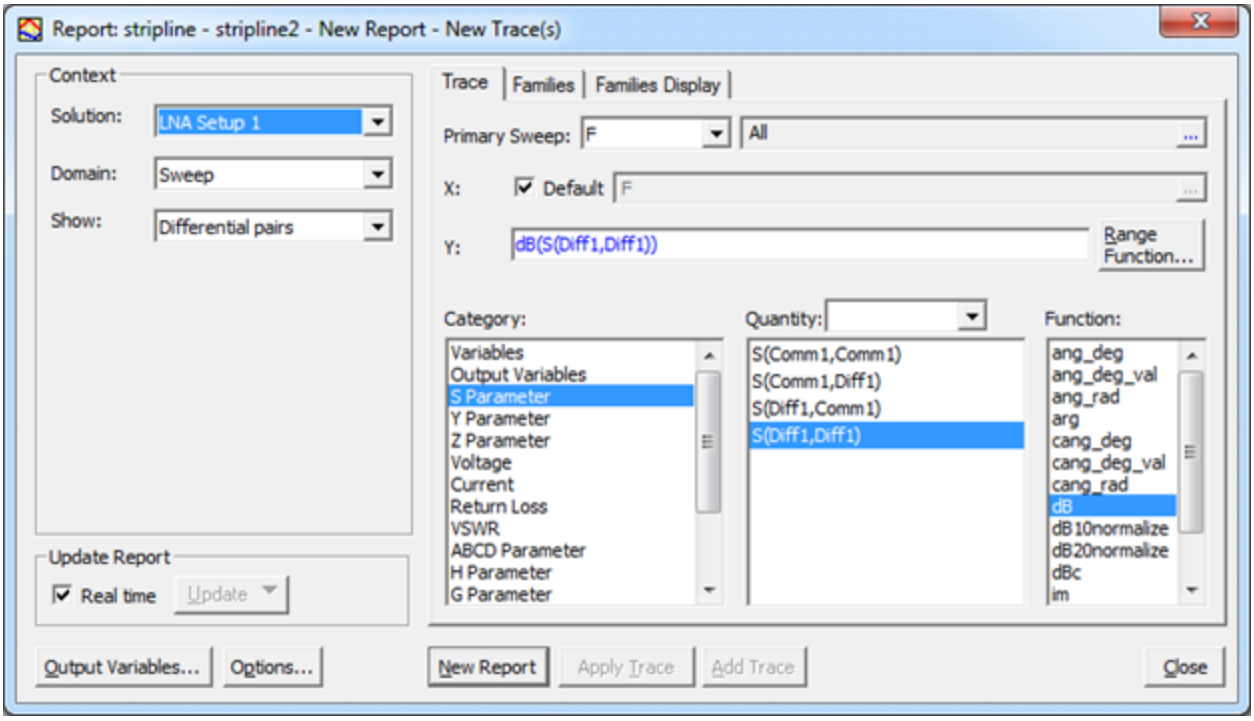
Differential Pairs window

You can set differential pairs for the Interface Ports through the Differential Pairs window. The values can then be accessed in the Reporter window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



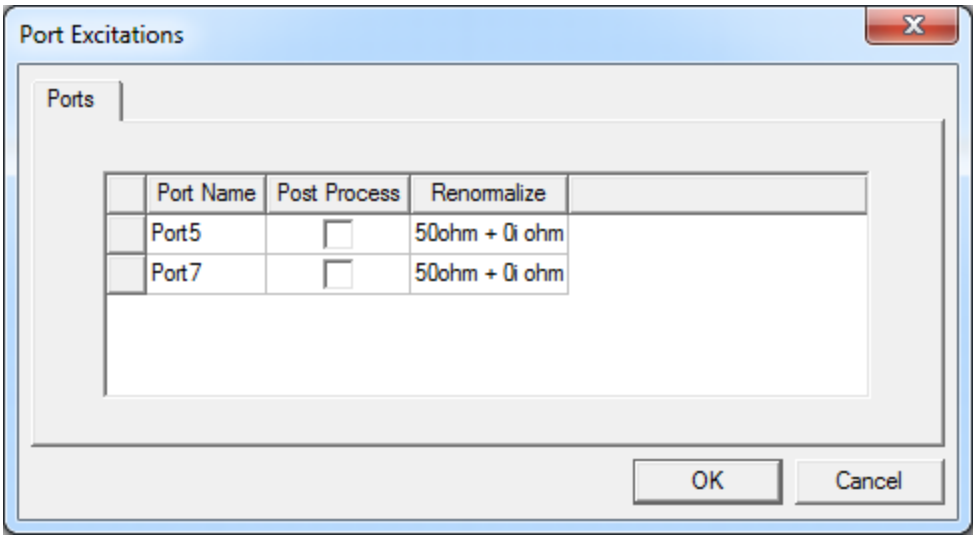
Here is the Reporter window with the differential quantities.



Port Excitations 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 set the renormalization impedance for the Interface Ports using the Port Excitations window.



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**.

PinToPin Utility

The PinToPin automation module can be used to quickly extract Ansys HFSS™ high-frequency electromagnetic simulation software or Ansys SIwave™ printed circuit board (PCB) and package electromagnetics simulation software models for specified nets of package and PCB geometries. It is intended to establish a repeatable process for robust geometry extraction, port configuration, and passive model assignment. Process execution can be driven using a minimal GUI interface, or non-graphically from an XML template. The data flow is modular, created 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 Ansys Electronics Database (EDB) .Net API and can be customized or extended to satisfy specific and unique modeling requirements.

The PinToPin Utility contains the following:

- Command Line
- Linux Compatibility
- Windows Utility
- XML Interface
- Environment Variables

Launching PinToPin Utility

To launch the PinToPin utility in non-graphical mode, type:

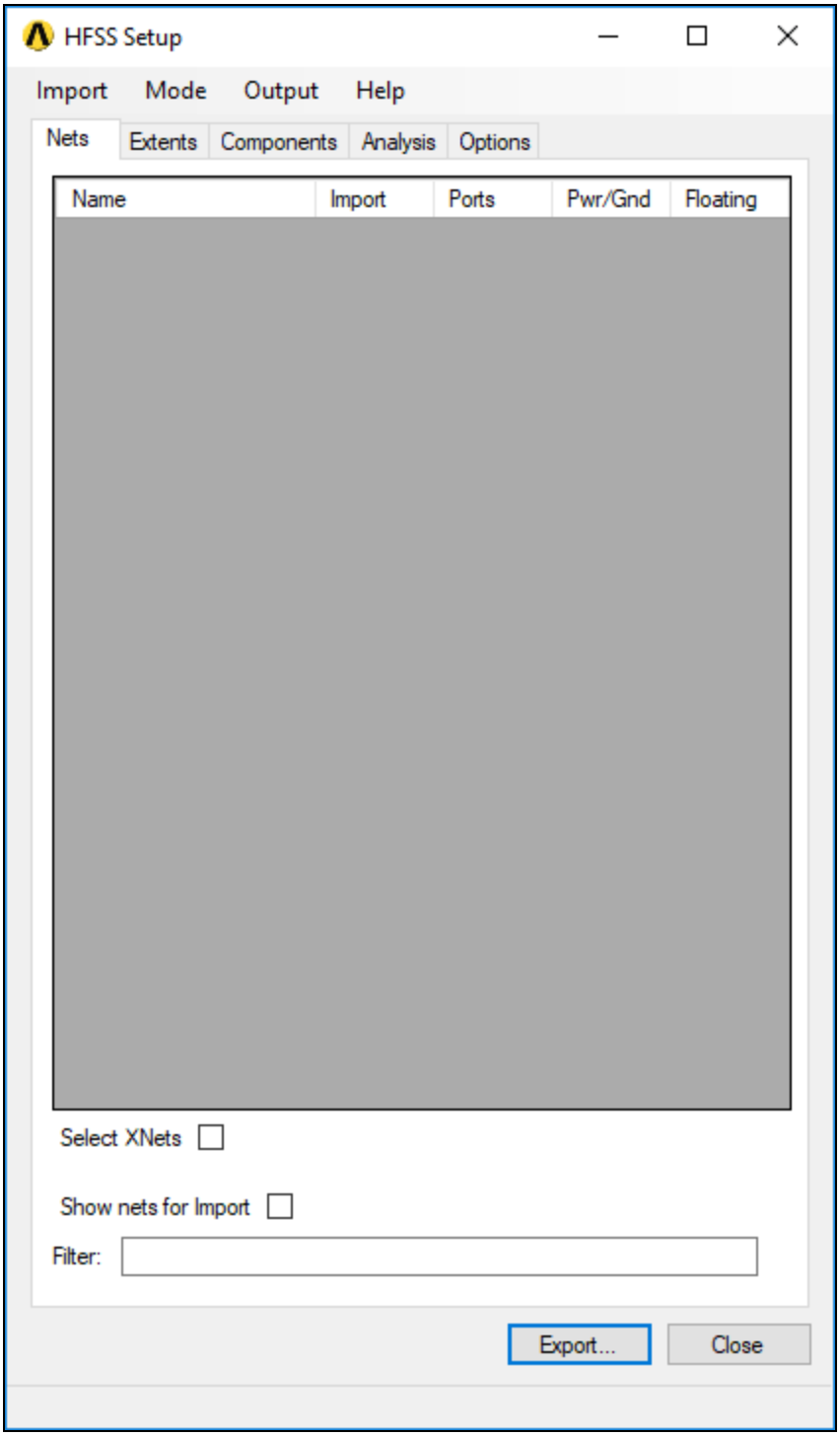
- **PinToPinSetup** <workingDir> <path to XML control file> <path to selected EDB or XML output>

To launch the PinToPin utility from within the Ansys Electronics Desktop (AEDT)™ electronics systems design platform:

- From the **Automation** tab on the ribbon, click **PinToPin Setup**

The PinToPin utility launches in a window labeled **HFSS Setup**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Importing a File

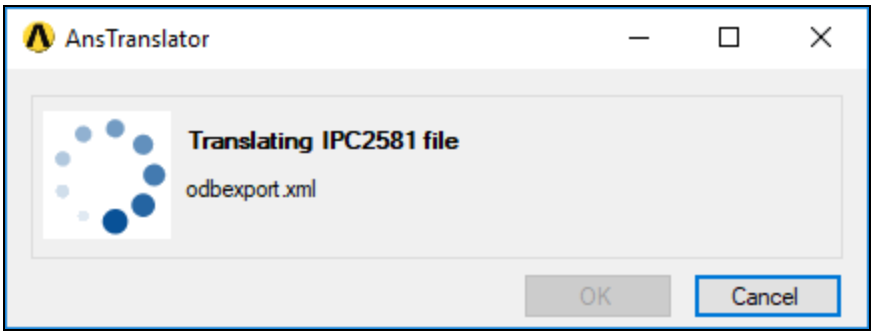
To import a file into PinToPin utility, click **Import** and select a file type.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

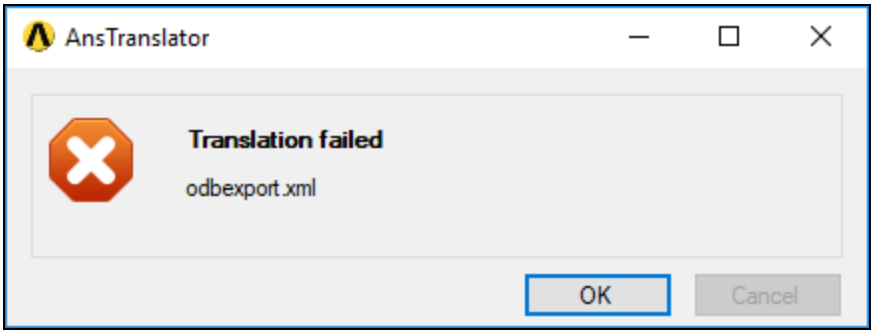
Supported file types are:

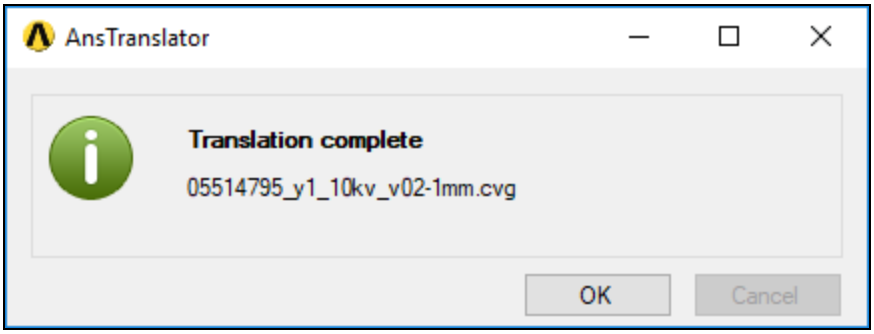
- Cadence Allegro/APD/SIP (*.brd, *.mcm, *.sip)
- ODB++ (*.tgz)
- IPC2581 (*.xml, *.cvg)
- Ansys EDB (*.def)
- XML configuration file (*.xml)

For Cadence, ODB++, and IPC2581 files, the **AnsTranslator** window appears and alert you of translation progress:



It then lets you know whether translation has succeeded or failed.



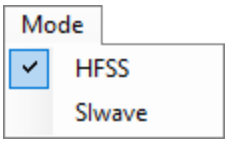


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once the file has been loaded, the window name changes to the file name.

Selecting a Simulation Mode

Click **Mode** and select **HFSS** or **SIwave**.



The contents of the **Analysis** tab change based on this selection, and the **Extents** tab appears only for HFSS mode.

Exporting a File

Click **Output** and select one of the following:

- **HFSS 3D Layout** – *.aedb folder that can be imported into either AEDT or SIwave.
- **XML** – *.xml file that can be used in subsequent calls to PinToPin. This is a good way to save a configuration or create a template that can be called repeatedly to create multiple models on the same file.

Click **Export** to generate the selected file type.

Troubleshooting PinToPin Utility

PinToPinSetup writes log files to the working directory. These log files may provide useful information about warnings or errors encountered during model generation.

PinToPin Command Line

`PinToPinSetup` <workingDir>

Launch the utility in graphical mode. Windows support only.

- <workingDir>: path to the working directory. If not selected by the user, the working directory are the first path found of:
    1. The path specified by the TMP environment variable.
    2. The path specified by the TEMP environment variable.
    3. The path specified by the USERPROFILE environment variable.
    4. The Windows directory.

`PinToPinSetup` <workingDir> <inputXML> <output>

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Launch the utility in non-graphical mode. Windows and Linux support.

- <inputXML>: path to the input XML control file.
- <output>: path to the selected Ansys EDB or XML control file output.

The ANSYSEM_INSTALL_DIR environment variable must also be set for PinToPin to locate required translators and EDB assemblies.PinToPinSetup is release specific, so this directory must be set consistent with the executing assemblies. The Release version of the PinToPinSetup assembly can be found in the **Help** > **About** menu.

PinToPinSetup and AnsTranslator write log files to <workingDir>. These log files may provide information on model configuration, and any warnings or errors encountered during model generation.

To call .mcm and .brd files and merge them together in -ng mode, e.g., BAT file:

```
set TESTDIR=%PROJDIR%\Project_Folder
set EDBOUT=%TESTDIR%\_working\merge.aedb
call :TRANSLATE %TESTDIR% PCB_file PKG_file
"%ANSYSEM_INSTALL_DIR%\common\IronPython\ipy64.exe"
"%PROJDIR%\package_merge.py" "%EDBFN1%"
"%EDBFN2%" "%EDBOUT%" -refdesTo U1 -refdesFrom B1
"%ANSYSEM_INSTALL_DIR%"\ansysedt.exe -scriptargs
"%EDBOUT%\layout.def" -runscript
"%BASEDIR%\PinToPinSetup__ImportEdbAndFinishSetup.py" -logfile
ImportEDB.log
EXIT /B
```

To call non-graphically and export the basic XML template, e.g., BAT file

```
set BASEDIR=D:\FolderNaAME
set PROJDIR=%BASEDIR%\TEST\XMLexport
"%BASEDIR%\PinToPinSetup.exe" "%PROJDIR%\_tmp"
"%PROJDIR%\Extract_default.xml"
"%PROJDIR%\base.xml"
EXIT /B
```

PinToPin Linux Compatibility

Only non-graphical execution is supported for Linux.

PinToPinSetup.exe is a .Net assembly and requires indirect execution through Mono, which is installed as part of the Ansys Electronics Desktop (AEDT)™ electronics systems design platform application. This is typically accomplished using a PinToPinSetup wrapper:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
#! /bin/bash

export ANSYS_OADIR="${ANSYSEM_INSTALL_DIR}/common/oa"

export PATH="${ANSYSEM_INSTALL_
DIR}/common/mono/Linux64/bin:${ANSYSEM_INSTALL_DIR}:${ANSYSEM_
INSTALL_
DIR}/Help:/bin:/usr/bin:/usr/kerberos/bin:/usr/local/bin:/usr/atria/b
in:${PATH}"

export LD_LIBRARY_PATH="${ANSYSEM_INSTALL_DIR}:${ANSYSEM_INSTALL_
DIR}/common/Framework/bin/Linux64:${ANSYSEM_INSTALL_
DIR}/common/oa/lib/Linux64/opt:${ANSYSEM_INSTALL_
DIR}/EKM/programs/jre1.7.0_07/lib/amd64/server:${ANSYSEM_INSTALL_
DIR}/EKM/EKM-Connector/bin/Linux64:${ANSYSEM_INSTALL_
DIR}/common/mono/Linux64/lib:${ANSYSEM_INSTALL_
DIR}:/usr/lib:/usr/lib64:/lib:/lib64:/usr/lib/tls:/usr/lib64/tls:${AN
SYSEM_INSTALL_DIR}/em64t/bin:${ANSYSEM_INSTALL_DIR}/ia32/bin"

mono ${ANSYSEM_INSTALL_DIR}/PinToPinSetup.exe $1 $2 $3 4
```

PinToPin Window Utility

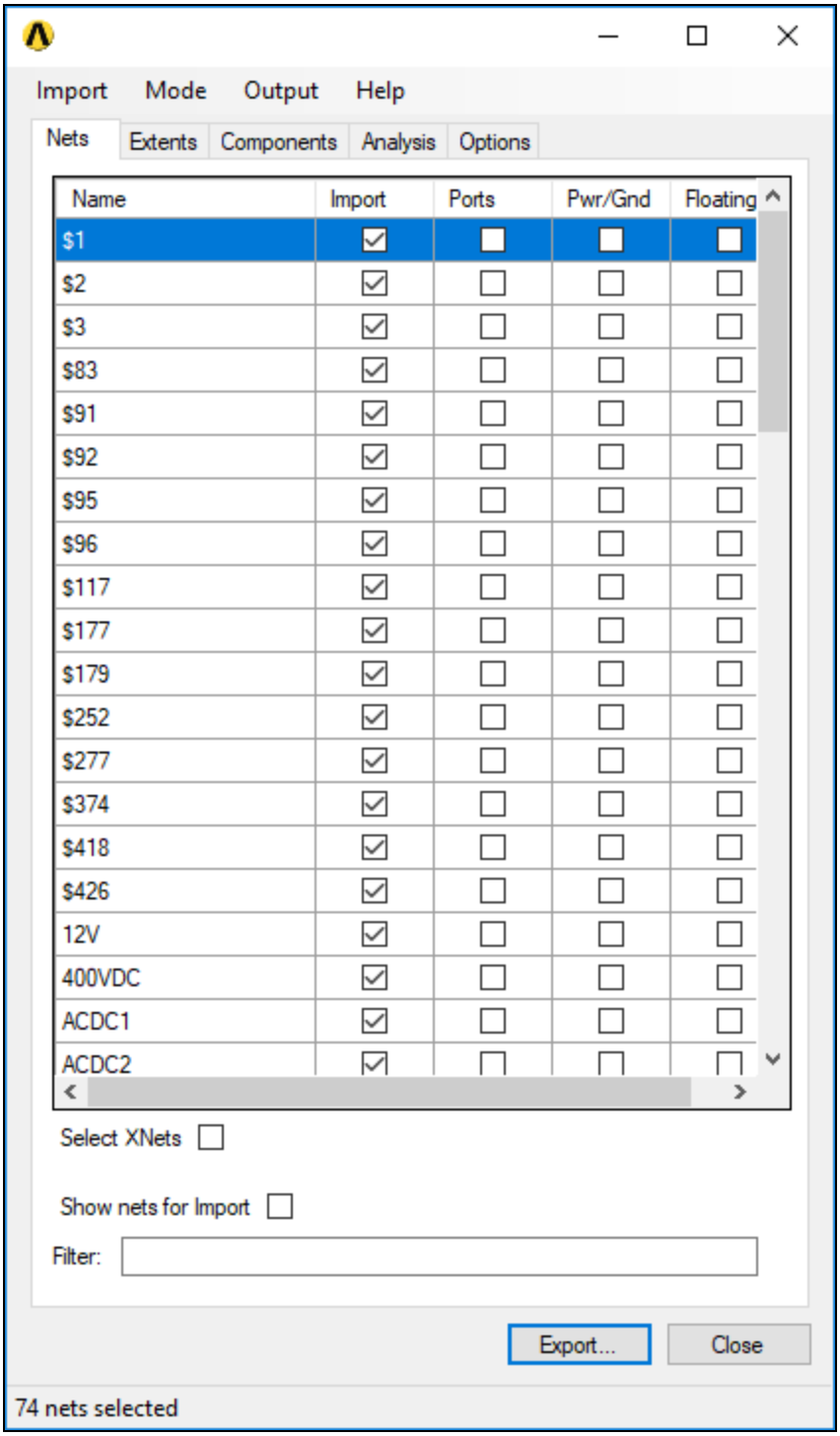
The PinToPin utility contains up to five tabs. Click a tab's name to learn more:

- **Nets**
- **Extents**
- **Components**
- **Analysis**
- **Options**

Nets Tab

The **Nets** tab allows you to select nets for import, and quickly denote ports, power/ground nets, and floating nets. Also filter the nets using regular expressions.

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 of this tab shows the number of nets currently selected.

Options include:

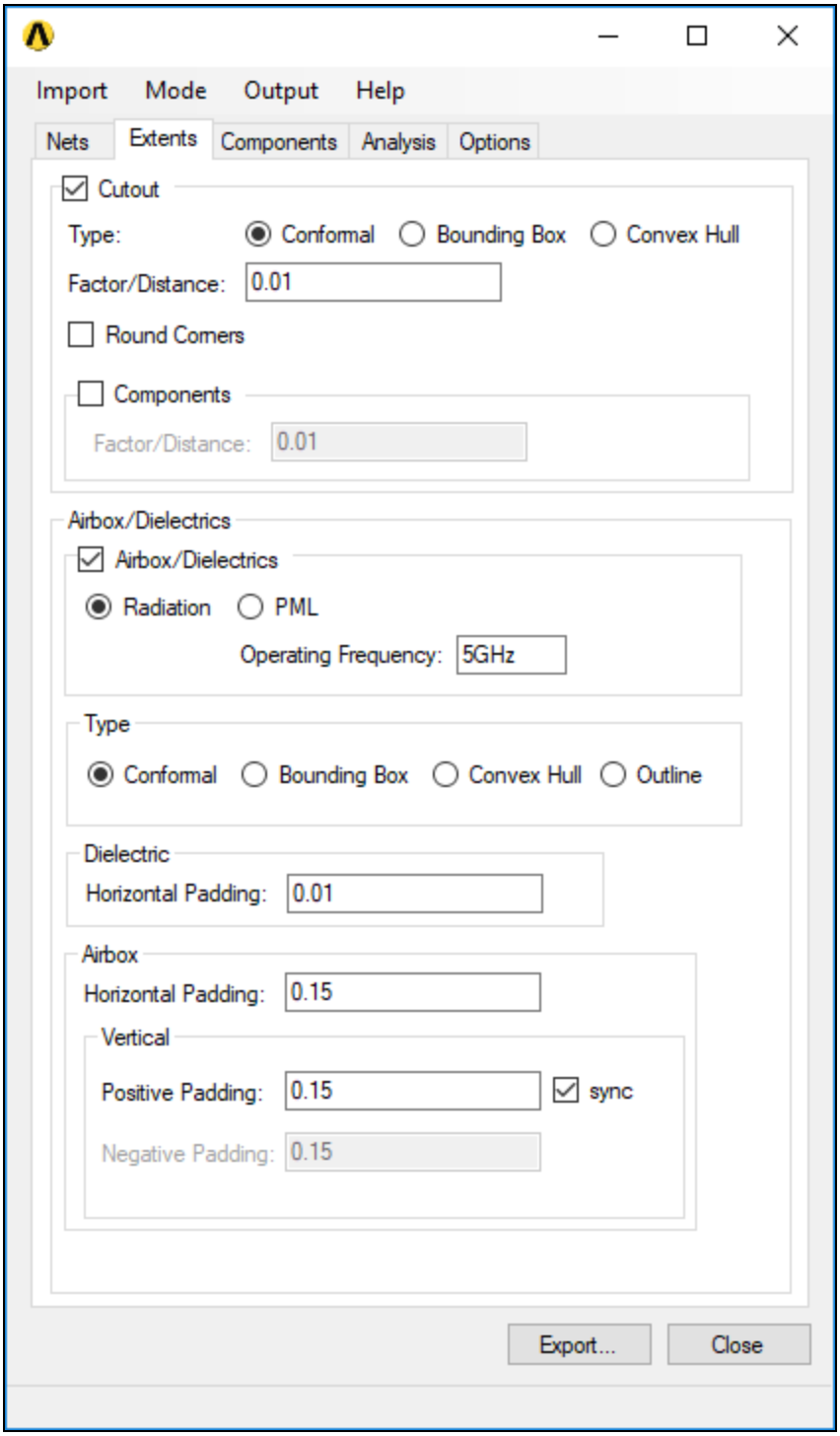
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Import** – use the check boxes to select nets for import.
- **Ports** – use the check boxes to identify nets for which ports are created at component pins of class IO, IC, or Other.
- **Pwr/Gnd** – use the check boxes to identify power or ground nets. When clipping the model region, nets in this class are not used to calculate the boundary. Nets included but not assigned Pwr/Gnd are preserved by the clipping operation.
- **Floating** – use the check boxes to identify floating power or ground nets. Nets assigned as floating are not tied into reference planes at component interfaces.
- **Select XNets** – when this option is enabled, selecting a net also includes any extended nets. Nets are in the same extended net if they have a serial connection through an R, L, or C component.
- **Show nets for Import** – when this option is enabled, the net list displays only nets selected for inclusion.
- **Filter** – enter regular expressions in this field to search for nets. Only those that satisfy the regular expression are shown.

Extents Tab

The **Extents** tab allows you to specify cutout, airbox, and dielectrics settings. It appears only in **HFSS** mode.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



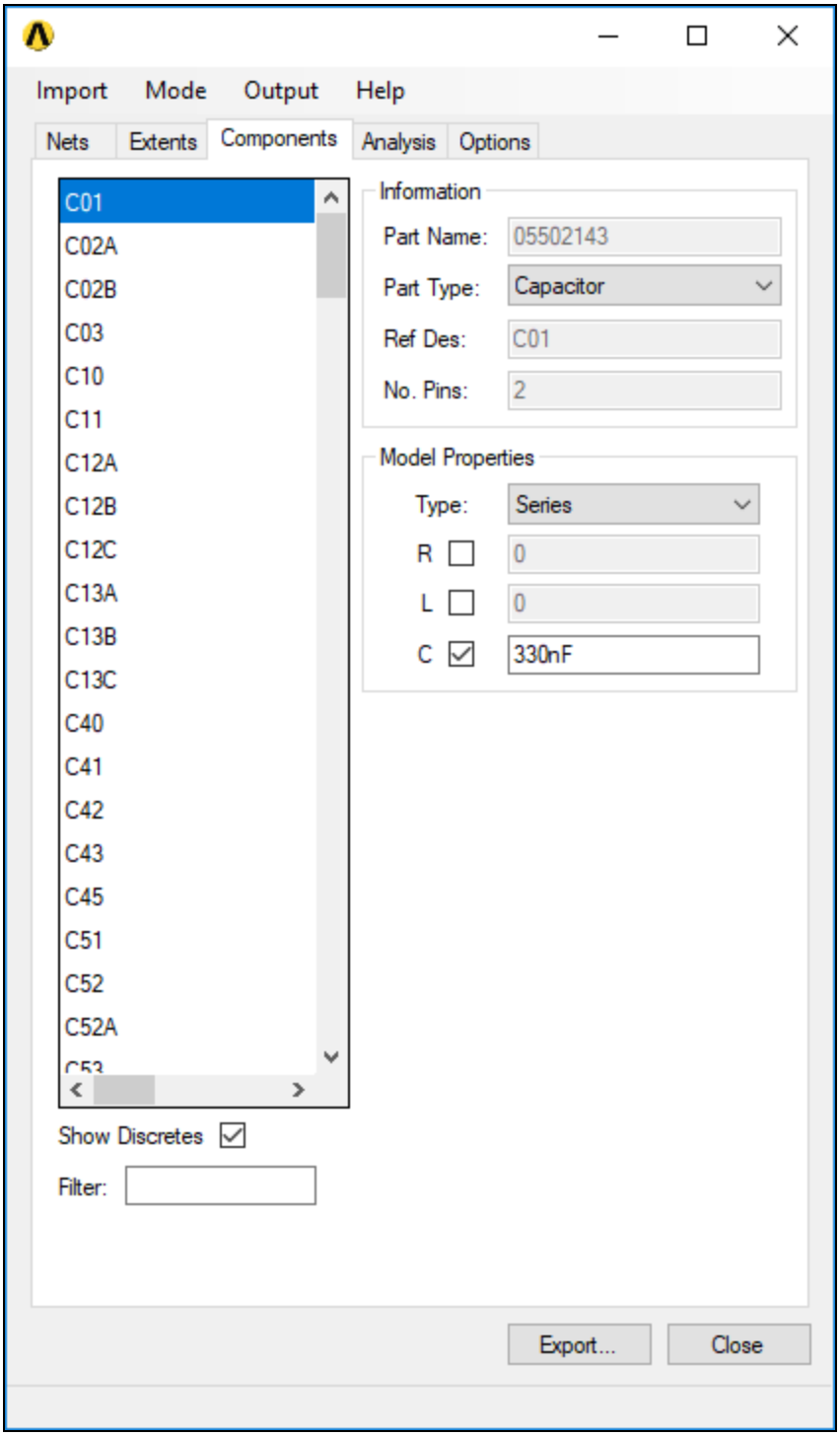
For more information about extent settings, see: HFSS 3D Model Extents.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Components Tab

The **Components** tab allows you to change component part types, and set model, solderball, die, and port properties.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

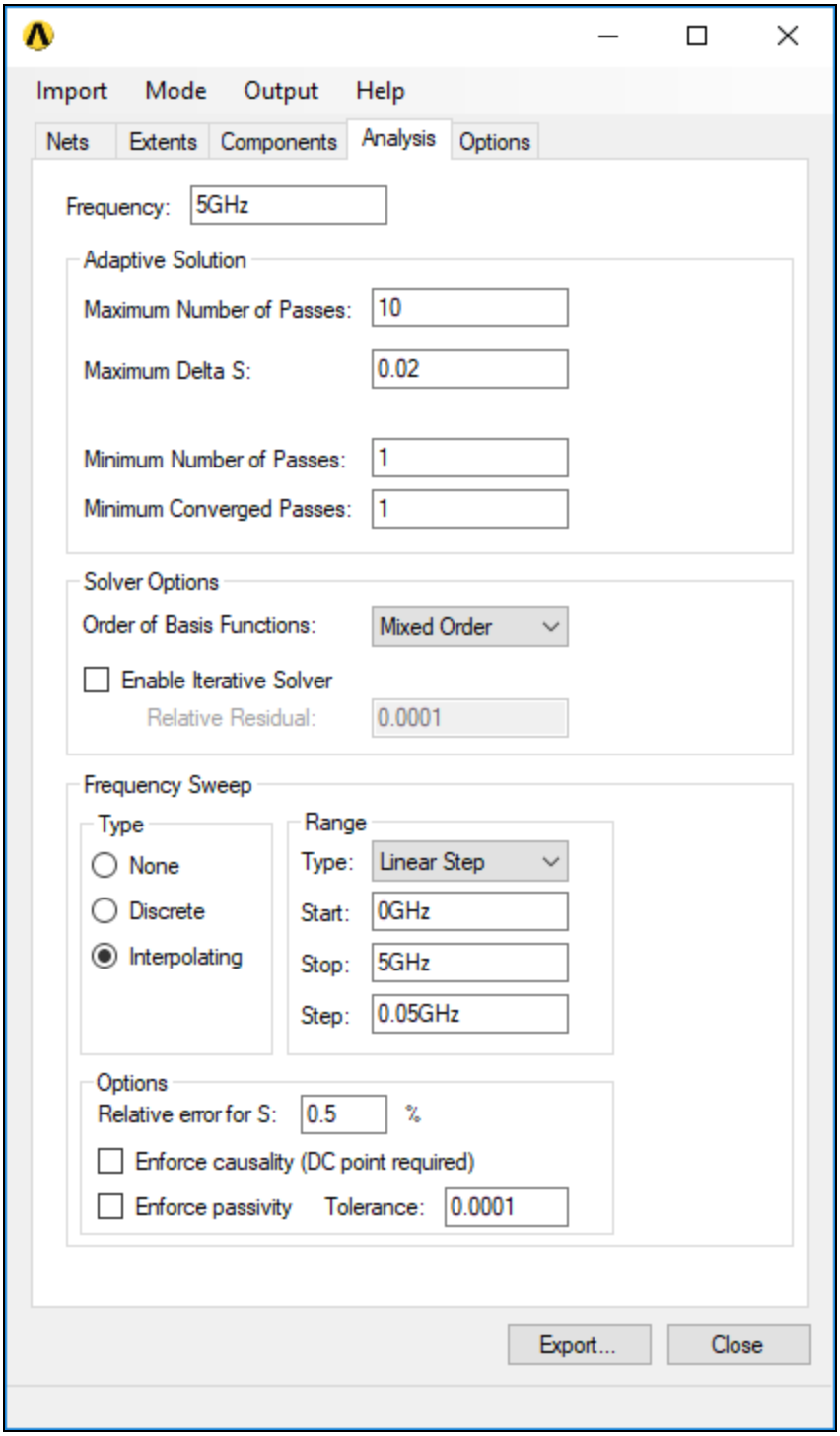
- **Show Discretes** – enable to display resistor, capacitor, and inductor component types.
- **Filter** – enter regular expressions in this field to search for components. Only those that satisfy the regular expression are shown.

Select a component from the list at the left to view its **Information** and **Properties**. Options vary based on the **Part Type**, which change. Generally, you can alter model properties, die properties, solderball properties, and port properties.

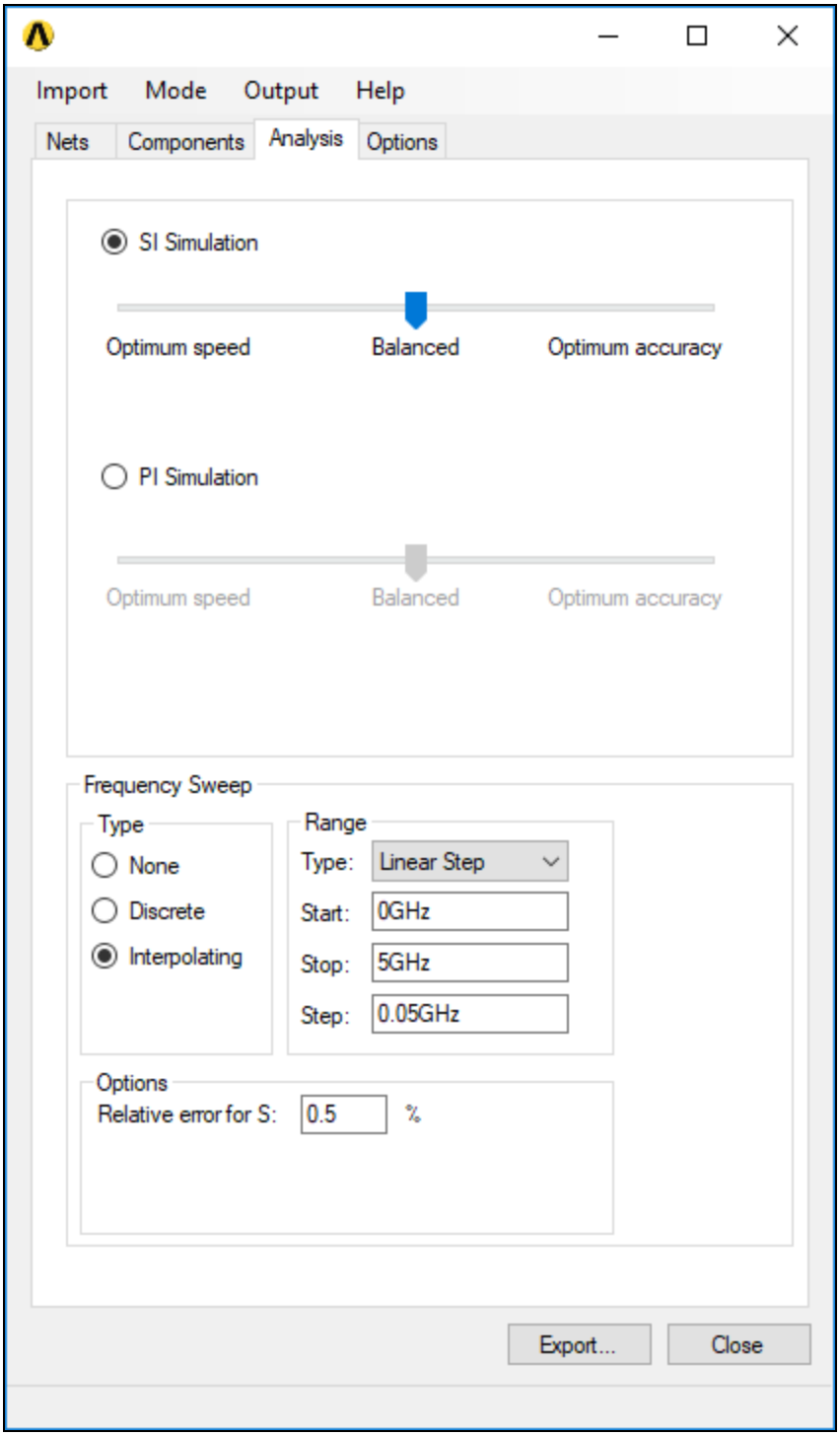
Analysis Tab

The **Analysis** tab allows you to set simulation and frequency sweep options. Options vary based on whether you are in **HFSS** or **SIwave** mode.

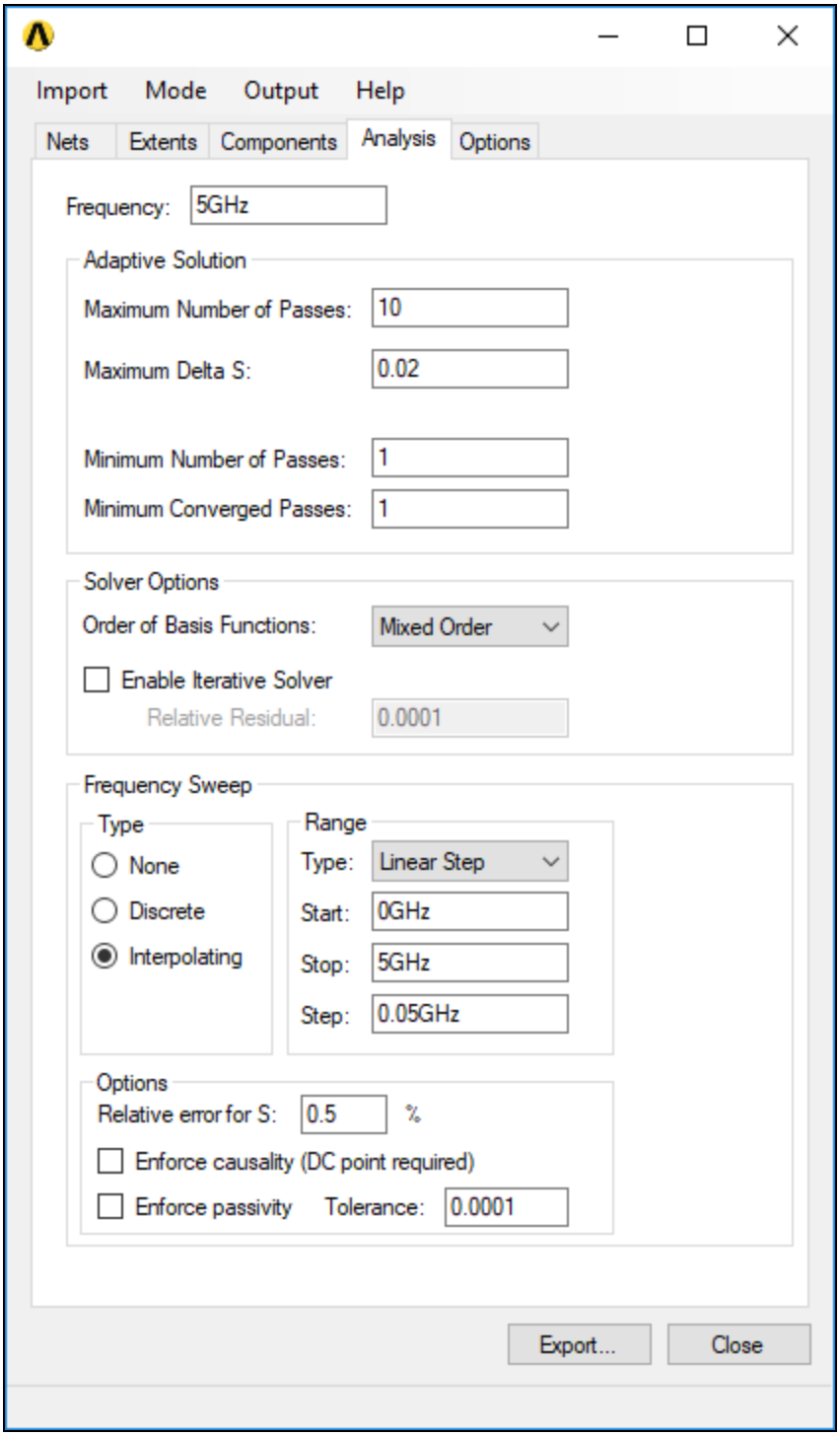
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



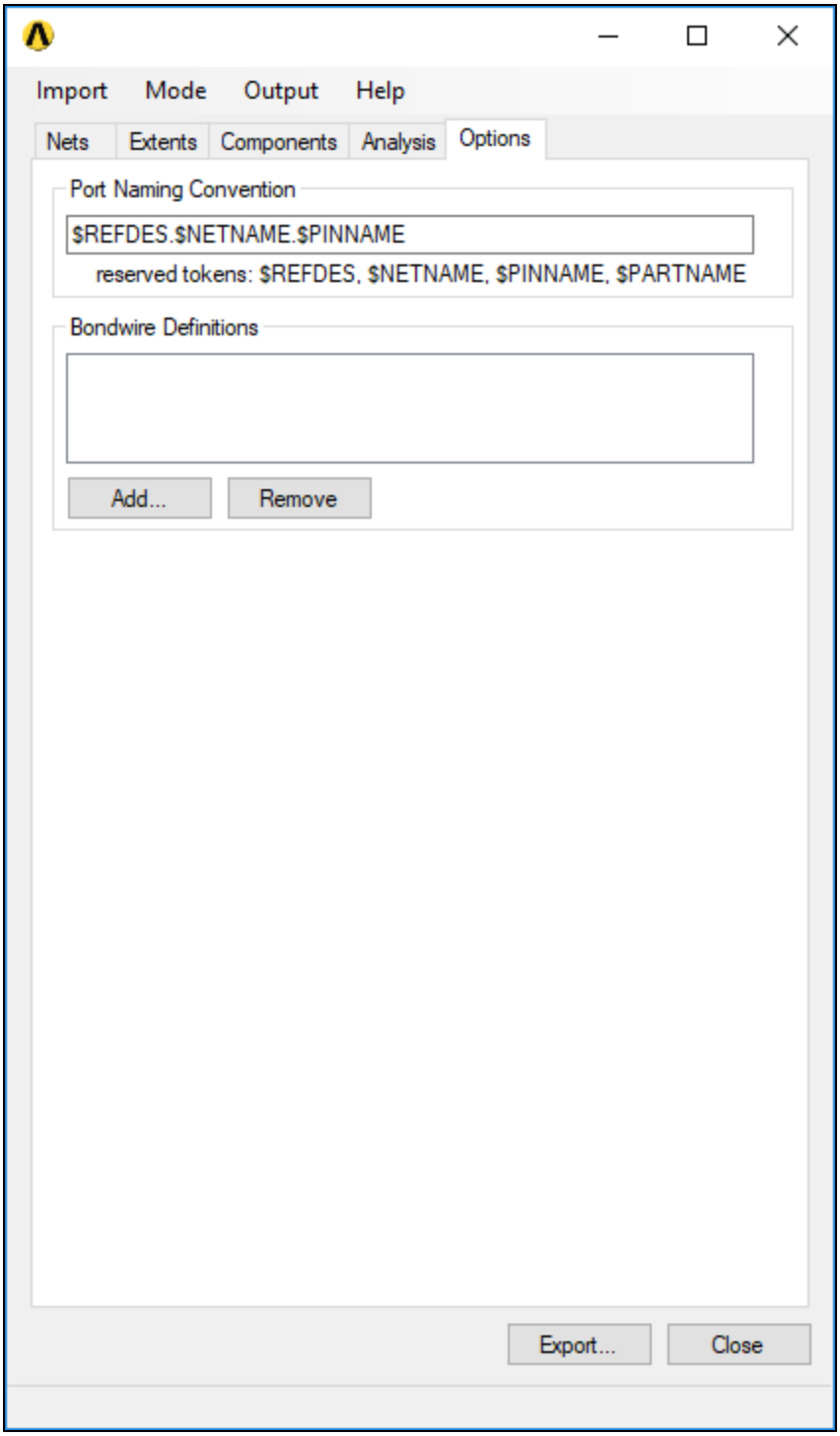
For information about setting up an Ansys SIwave™ simulation, consult the Ansys SIwave™ online help.

For information about setting up an Ansys HFSS™ simulation, see: Adding a 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.

Options Tab

The **Options** tab allows you to alter the port naming convention or bondwire definitions.



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 **Port Naming Convention** field, use the reserved tokens to designate a port naming convention. For example, entering `$REFDES_$PINNAME_$NETNAME` results in ports such as `U37_A10_DQ7`.

From the **Bondwire Definitions** area, click **Add** to upload an XML bondwire definition file.

PintoPin XML Interface

An XML interface is available for complete non-graphical execution of PinToPin. This enables rapid and repeatable model creation from previously captured “template” XML files. Additionally, XML documents are easily generated by most scripting languages, simplifying the integration of PinToPin in external applications.

For simplicity, the XML schema is included as an embedded resource in the Ansys.Ansoft.EdbBuilderUtils.dll assembly. Upon request, it can be provided as a stand-alone xsd file. Rather than focus on schema, this document instead describes common use cases of the XML interface.

- Basics
- Stackup Modification
- Component Models
- Pin Groups
- Surface Roughness

Basics

The easiest way to generate a comprehensive, design-specific XML document is to translate and export to XML using the GUI, or to translate and export XML on the command-line:

```
PinToPinSetup translate.xml my_design.xml
```

Where translate.xml is:

```
<?xml version="1.0"encoding="utf-8"standalone="yes"?><c:Control
SourceDataFileName="C:\fairbairn_new_DBdoctor165_
BB.brd"xmlns:c="http://www.ansys.com/EDBBuilderUtils/SetupInfo"/>
```

Note the SourceDataFileName attribute provides the path to the input for PinToPin. All non-graphical XML execution requires this be provided.

Each XML document also allows for the following elements:

Options

Optionally specify the PortNamingConvention and ImportXNets option:

```
<Options>
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  <PortNamingConvention Value="$REFDES.$NETNAME"/>
  <ImportXNets>False</ImportXNets>
</Options>
```

Stackup

The optional Stackup element includes material definitions and layers. It may be used to re-define materials or to change parameters in a stackup, as described in a later section.

Nets

Specify the net configuration. If a net is not included in the XML, it is not imported (unless captured by the ImportXNets option). Pwr/Gnd nets must have the Class=”Power/Ground” attribute:

```
<Nets>
  <Net Name="B-MII-RXD0" PinsBecomePorts="true" Import="True"/>
  <Net Name="B-MII-RXD1" PinsBecomePorts="true" Import="True"/>
  <Net Name="GND" Class="Power/Ground" Import="True"/>
</Nets>
```

CutoutSubdesign

If applicable, describe the clipping polygon for the design. Note the Clip element should always reference NetClass=”Power/Ground”.

```
<CutoutSubdesign>
  <Auto Type="Conformal" Expansion="0.01"oundCorners="False"/>
  <Clip NetClass="Power/Ground"/>
</CutoutSubdesign>
```

Definitions

Specify bondwire and part definitions:

```
<Definitions>
  <CDNWirebond File="C:\Ansoft_Wire_Profiles.xml"/>
  <PartMap>
    <Part Name="ABC123">
      <NPortModel Type="Touchstone" FileName="C:\test.s2p"/>
    </Part>
  </PartMap>
</Definitions>
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundaries

Define the model extents:

```
<Boundaries>
  <Extents Type="bbox" DielHorizFactor="0" AirboxHorizFactor="0.15"
  AirboxVertFactorPos="0.15" AirboxVertFactorNeg="0.15"
  UseRadiationBoundary="True"/>
</Boundaries>
```

Components

This element is used to specify the component configurations. When exporting from PinToPin, itcontains every component in the layout. However, only those components requiring solderballs or a model other than their default need be included. Many of the attributes are included to enable GUI execution directly from an XML file but are not requiredfor non-graphical model creation.

```
<Components schemaVersion="1.0">
  <Component RefDes="U2" PartName="SQFP28X28_208" PartType="IC">
    <DieProperties Type="Flip chip" Orientation="Chip down"
    Height="0"/>
    <SolderballProperties Shape="Cylinder" Diameter="168um"
    MidDiameter="168um" Height="279um"/>
    <PortProperties ReferenceOffset="0" ReferenceSizeAuto="True"
    ReferenceSizeX="0" ReferenceSizeY="0"/>
  </Component>
  <Component RefDes="C1" PartName="CC0603" PartType="Capacitor"
  NumPins="2">
    <DieProperties Type="None" Orientation="Chip up" Height="0.0"/>
    <SolderballProperties Shape="None" Diameter="610um"
    MidDiameter="610um" Height="1.02mm"/>
    <PortProperties ReferenceOffset="0.0" ReferenceSizeAuto="True"
    ReferenceSizeX="" ReferenceSizeY=""/>
    <RLCModel Type="Series" C="10uF"/>
  </Component>
...
```

A minimal amount of information can be provided to enable solderball creation on a component. For example:

```
<Component RefDes="U2" PartType="IC">
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  <SolderballProperties Shape="Cylinder" Height="333um"/>
</Component>
```

Remaining parameters uses default values.

Setup

One or more simulation Setups can be provided. The Setup element is used to describe simulation parameters and frequency sweeps for Ansys HFSS™ analysis. The SIwaveSetup element is used to describe simulation parameters and frequency sweeps for Ansys SIwave™ printed circuit board (PCB) and package electromagnetics simulation software analysis.

```
<Setup schemaVersion="1.0">
  <Fadapt>5GHz</Fadapt>
  <MaxNumPasses>10</MaxNumPasses>
  <MaxMagDeltaS>0.02</MaxMagDeltaS>
  <MinNumPasses>1</MinNumPasses>
  <MinConvergedPasses>1</MinConvergedPasses>
  <BasisOrder>Mixed</BasisOrder>
  <UseIterativeSolver>False</UseIterativeSolver>
  <IterativeSolverRelativeResidual>0.0001</IterativeSolverRelativeRes
  idual>
  <CurveApproxSettings>
    <ArcAngle>12deg</ArcAngle>
    <StartAzimuth>0</StartAzimuth>
    <MaxArcPoints>10</MaxArcPoints>
    <UseArcToChordError>True</UseArcToChordError>
    <ArcToChordError>10um</ArcToChordError>
  </CurveApproxSettings>
  <FreqSweep Name="Sweep1" UseQ3DForDC="False">
    <Interpolating RelativeErrorS="0.5" UseErrorZ0="False"
    PercentErrorZ0="1" EnforceCausality="False"
    EnforcePassivity="False" PassivityTolerance="0.0001">
      <LinearStep Start="0" Stop="5000000000" Step="50000000"/>
    </Interpolating>
  </FreqSweep>
</Setup>


<SIwaveSetup Name="SIwave Setup 1">
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  <SISimulation>Balanced</SISimulation>
  <FreqSweep Name="TestSweep1" ComputeExactDCPoint="True"
  Use3DDDM="False">
    <Interpolating RelativeErrorS="0.2">
      <LinearStep Start="0" Stop="10e9" Step="50e6"/>
    </Interpolating>
  </FreqSweep>
</SIwaveSetup>
<SIwaveSetup Name="SIwave Setup 2">
  <PISimulation>OptimumAccuracy</PISimulation>
  <FreqSweep Name="Sweep1" Use3DDDM="True">
    <Interpolating>
      <DecadeCount Start="1e3" Stop="10e9" Count="10"/>
    </Interpolating>
  </FreqSweep>
</SIwaveSetup>
```

Stackup Modification

Often, 3rd-party data does not contain material definitions and stackup suitable for simulation. In this case, the Stackup element can be used to make modifications.

Material definitions may be overridden by name:

```
<Material Name="FR-4">
  <Permittivity>
    <Double>4.5</Double>
  </Permittivity>
  <Permeability>
    <Double>1</Double>
  </Permeability>
  <Conductivity>
    <Double>0</Double>
  </Conductivity>
  <DielectricLossTangent>
    <Double>0.002</Double>
  </DielectricLossTangent>
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
</Material>
```

New materials can also be defined, then referenced in the stackup.

The Stackup/Layer element may be used to change specific attributes of a named layer, for instance to change the thickness of layer TOP:

```
<Layer Name="TOP" Type="conductor" Thickness="1.4"/>
```

Material, Fill Material, Thickness, RoughnessVal, and Color can all be changed.

It is not appropriate to know the stackup names to change properties. If Layer names are omitted, the Stackup are mapped by layer type, with layer properties modified for the first matching sequence of layers:

```
<Layers LengthUnit="mil">
  <Layer Material="AIR" Thickness="2" Type="dielectric"/>
  <Layer FillMaterial="AIR" Material="COPPER" Thickness="1.5"
  Type="conductor"/>
  <Layer Material="FR-4" Thickness="7" Type="dielectric"/>
  <Layer FillMaterial="FR-4" Material="COPPER" Thickness="1.4"
  Type="conductor"/>
  <Layer Material="FR-4" Thickness="18" Type="dielectric"/>
  <Layer FillMaterial="FR-4" Material="COPPER"
  Thickness="1.4"Type="conductor"/>
  <Layer Material="FR-4" Thickness="7" Type="dielectric"/>
  <Layer FillMaterial="FR-4" Material="COPPER" Thickness="1.4"
  Type="conductor"/>
  <Layer Material="FR-4" Thickness="18" Type="dielectric"/>
  <Layer FillMaterial="FR-4" Material="COPPER" Thickness="1.4"
  Type="conductor"/>
  <Layer Material="FR-4" Thickness="7" Type="dielectric"/>
  <Layer FillMaterial="AIR" Material="COPPER" Thickness="1.5"
  Type="conductor"/>
  <Layer Material="AIR" Thickness="2" Type="dielectric"/>
</Layers>
```

Finally, layers can be deleted on the design altogether. This can be useful when trying to reduce model complexity:

```
<Layers>
  <Layer Name="UNNAMED_000" Delete="True"/>
  <Layer Name="TOP" Delete="True"/>
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  <Layer Name="UNNAMED_002" Delete="True"/>
  <Layer Name="RDL" Delete="True"/>
  <Layer Name="UNNAMED_004" Delete="True"/>
  <Layer Name="UNNAMED_006" Delete="True"/>
  <Layer Name="M1_TS" Delete="True"/>
  <Layer Name="UNNAMED_008" Delete="True"/>
  <Layer Name="UBM_BS" Delete="True"/>
  <Layer Name="UNNAMED_010" Delete="True"/>
</Layers>
```

Component Models

Resistor, Capacitor, and Inductor components can be assigned S-parameter, series or parallel RLC, and netlist models, specified at definition level as:

```
<Definitions>
  <PartMap>
    <Part Name="PartA">
      <NPortModel Type="Touchstone" FileName="TCM0403R-900.s4p"/>
    </Part>
    <Part Name="PartB">
      <RLCModel Type="Series" L="1111nH"/>
    </Part>
    <Part Name="PartC">
      <NetlistModel SolveIndependent="True">
                R1@ID %0 1_@ID 22.000000
                L1@ID 1_@ID %1 0.22n
                R2@ID %2 1_@ID 22.000000
                L2@ID 1_@ID %3 0.22n
      </NetlistModel>
    </Part>
  </PartMap>
</Definitions>
```

When a PartMap is provided, all component instances of a given part are assigned that model. Each component can override the PartMap of its definition by specifying a Model override. For example, a resistor of type “PartA” can use an alternate S-parameter model with an instance override:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<Component RefDes="R201" PartName="PartA" PartType="Resistor">
  <NPortModel Type="Touchstone" FileName="viawizard_
  HFSSDesign1.s4p"/>
</Component>
```

XML ENTITY references have been enabled to facilitate maintenance of Part definitions from external files. From the preceding example, this is accomplished by including the ENTITY reference as:

```
<?xml version="1.0" encoding="utf-8" standalone="no"?>
<!DOCTYPE doc[<!ENTITY partMapFile SYSTEM
"C:\Components\partmap.xml">]>
```

The Definitions element is then given as:

```
<Definitions>
  <PartMap>&partMapFile;</PartMap>
</Definitions>
```

Pin Groups

Pin Groups can be created for each component by referencing either Nets or explicit Pins. Once a Pin Group has been created, to use it in a Port definition, it must be referenced in a subsequent Port element. Various combinations of these constructs can be seen here:

```
<Component RefDes="U7" ...>
  ...
  <!--Inclusion by net-->
  <PinGroup Name="U7.GND_PG">
    <Net Name="GND"/>
  </PinGroup>
  <PinGroup Name="U7.VCC_PG">
    <Net Name="VCC"/>
  </PinGroup>
  <Port Name="U7.VCC" Z0="0.002">
    <PinGroup Name="U7.VCC_PG"/>
    <PinGroup Name="U7.GND_PG"/>
  </Port>
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  <!--Port between two Pin Groups -->
  <Port Name="U7.B-MII-RXD0" Z0="50">
    <Pin Name="131"/>
    <PinGroupName="U7.GND_PG"/>
  </Port>

  <!--Port between Pin and PG -->
  <Port Name="U7.B-MII-RXD1" Z0="50">
    <Pin Name="132"/>
    <PinGroup Name="U7.GND_PG"/>
  </Port>
</Component>
```

Surface Roughness

To define surface roughness models, include new xml elements under `Options`:

```
<c:Control xmlns:c=" http://www.ansys.com/EDBBuilderUtils/SetupInfo
">
  <Options>
    <PortNamingConvention Value="$REFDES.$NETNAME" />
    <DefaultRoughness RoughnessVal="0.51"
    HallHuraySurfaceRatio="2.91" />
    <DefaultTopRoughness RoughnessVal="0.52"/>
    <DefaultBottomRoughness RoughnessVal="0.53"
    HallHuraySurfaceRatio="2.93" />
    <DefaultInternalRoughness RoughnessVal="0.54" />
  </Options>
  <CutoutSubdesign>
    <Auto Type="BBox" Expansion="0.05" RoundCorners="False" />
...
```

All of these are optional. Application proceeds as follows:

```
IF layer has an explicit roughness model use it
ELSE
  IF layer is top-most conductor:
    IF DefaultTopRoughness is defined use it
    ELSE IF DefaultRoughness is defined use it
  ELSE IF layer is bottom-most conductor:
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    IF DefaultBottomRoughness is defined use it
    ELSE IF DefaultRoughness is defined use it
  ELSE
    IF DefaultInternalRoughness is defined use it
    ELSE IF DefaultRoughness is defined use it
```

PintoPin Environment Variables

ANSYSEM_INSTALL_DIR: This must be set and is used to locate the Ansys Electronics Desktop (AEDT)™ electronics systems design platform runtime and AnsTranslator.

ECAD_TRANSLATORS_INSTALL_DIR: If set, this path is given priority in finding AnsTranslator.

ANSOFT_EXTRACTA_MICROVIAS: If set to 1, modify existing definitions with a bounding-circle hole equal to the defined array's area. If set to 2, create a separate hole for each drill in the microvia array. AnsTranslator's default treatment of microvia arrays is to use a bounding-circle for the hole. ANSOFT_EXTRACTA_MULTIDRILL_CONSERVE_AREA can alternatively be set to configure the hole with equivalent area to the drill array.

ANSOFT_EXTRACTA_USE_COMPCLASS_IO_IC: If set, use defined Cadence COMP_CLASS to determine IO and IC components explicitly. Otherwise, infer from refdes and value properties.

PINTOPIN_USE_LUMPED_RLCPORT: For RLC components assigned port models, use a lumped port instead of a circuit port.

PINTOPIN_CREATE_SIWAVE_SETUP_FROM_HFSS_SETUP: If set, for every HFSS setup, create an Ansys SIwave™ setup with default SI settings and an equivalent frequency sweep.

PINTOPIN_DEFAULT_BONDWIRE_PROFILE: If set, use this profile definition name for all bondwire instances.

EDBBUILDERUTILS_XMLSCHEMA_REPORTWARNINGS: If set, report schema validation warnings encountered during valiation. Otherwise, XML schema is strictly enforced.

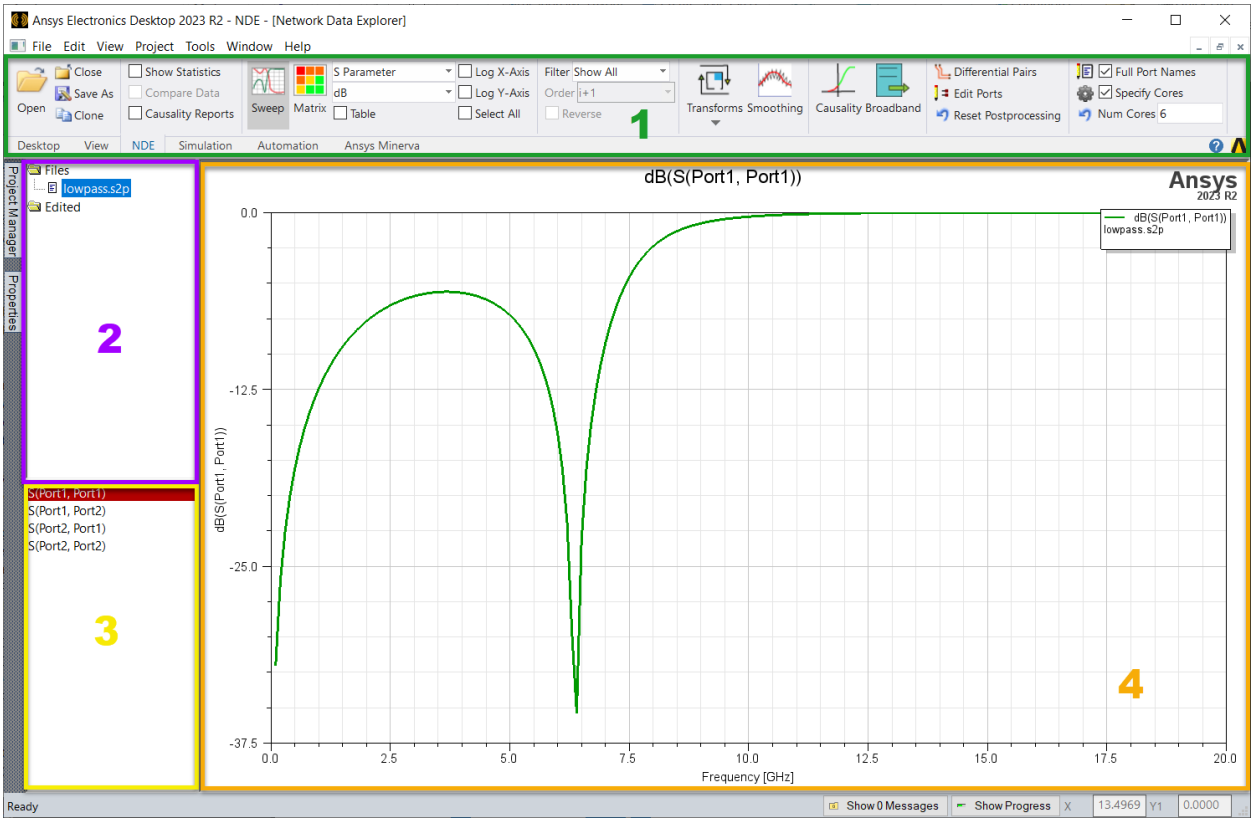
Network Data Explorer

The Network Data Explorer provides visualization, analysis, and manipulation tools for network data.

To access Network Data Explorer, click **Tools** > **Network Data Explorer**.

The **Network Data Explorer** 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.



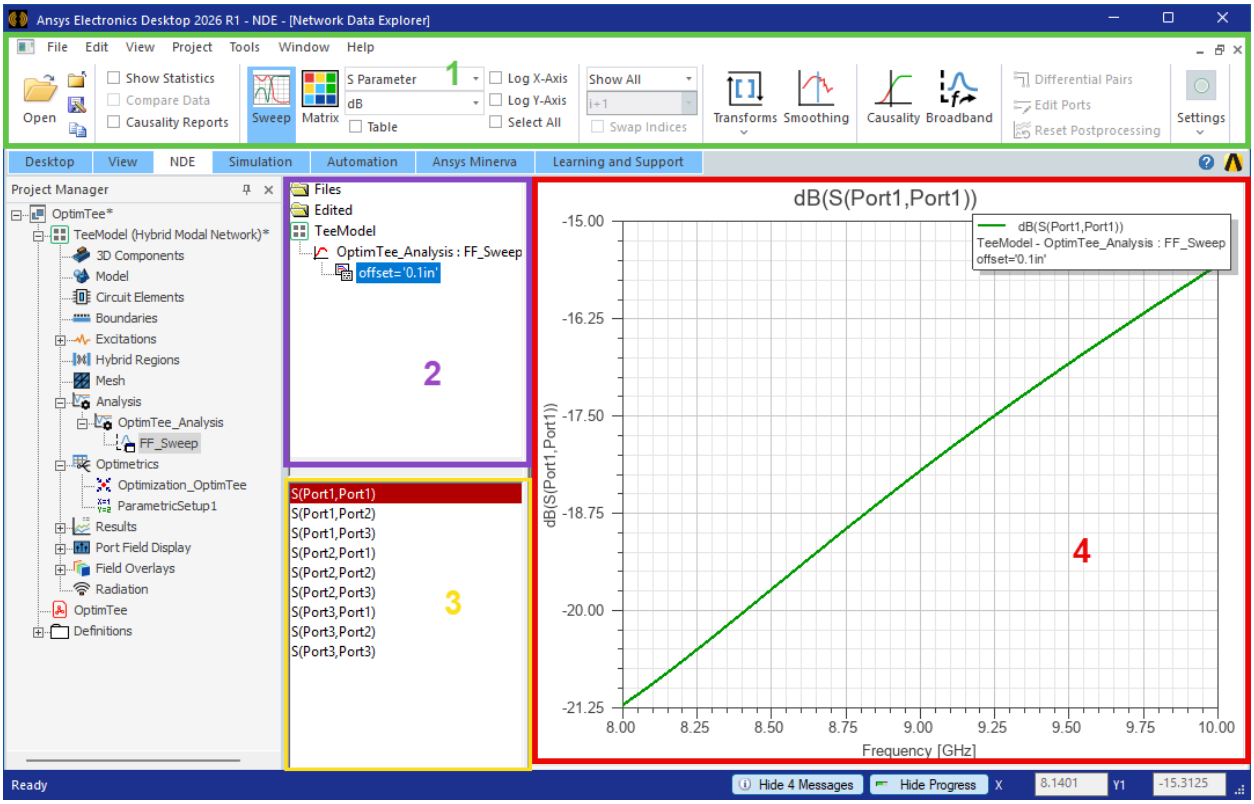
Network Data Explorer Overview

The Network Data Explorer window is divided into the following panes:

1. **NDE ribbon** – allows you to perform many functions of the Network Data Explorer.
2. **Network Data Selection pane** – allows you to select a network data file.
3. **Cell and Frequency Selection pane** – allows you to narrow your selection.
4. **Data View pane** – displays data in table or plot format.

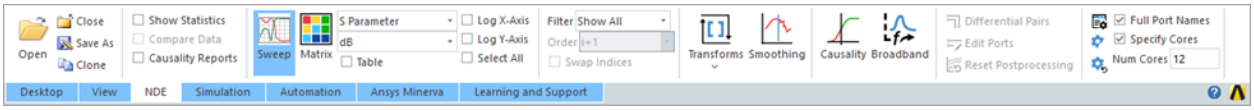
The panes are shown in the following figure. Additional information about each pane follows.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



NDE Ribbon

The ribbon provides access to many of the Network Data Explorer's functions and display options.



On this ribbon, you can control:

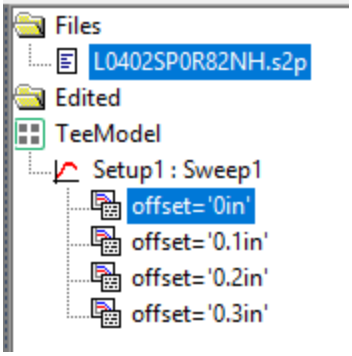
- Plotting - allows you to determine how the data is displayed.
- Quantity – allows you to select the type of quantity to display (parameter values, matrix statistics, or causality plots).
- Parameter Type – allows you to choose the parameter for display (S, Y, or Z parameters, Port Impedance, or Gamma).
- Format – allows you to decide the display function to apply to the data (e.g., magnitude, phase, dB, real, imaginary).

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 either SYZ data (*.s1p, *.ts, *.nmf, *.tab, *.m, *.cit) or Broadband data (*.sp).
- Check – allows you to check causality.
- Cores – allows you to enable or disable multithreading.
- Post-Process Selection – allows you to choose between Terminal Data and Differential Pairs, if your design includes Differential Pairs

Network Data Selection Pane

This pane allows you to view and compare various data sets. Original data sets appear under **Files**. Click one of these to see the data set as it was when it was opened. Altered data sets are listed under **Edited**. These data sets appear here when they have been smoothed, transformed, or changed in some way.



This pane also lists available variations for AEDT design solution data. Variations are listed under a setup name icon (for example “Setup1 : Sweep1”) that is listed under a design icon (“TeeModel”). Variations can be selected and displayed just like other data sets.

Use the **Shift** and **Ctrl** keys to select and display data sets from multiple files simultaneously. Multiple data sets can be selected to display multiple traces in sweep plots, but not in tables. Click the **Files** icon to select all data sets under **Files**. Click the **Edited** icon or a design/setup icon to do the same.

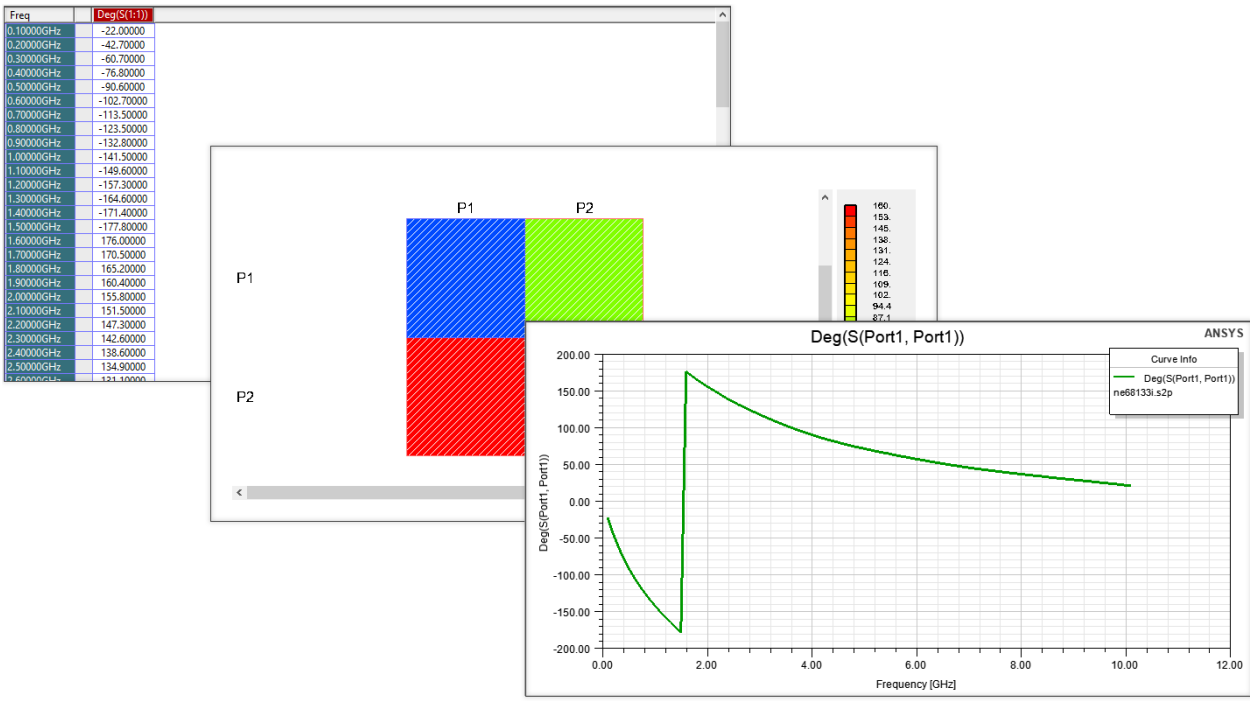
Cell and Frequency Selection Pane

Select the **Sweep** button on the NDE ribbon to display cell entries, for example S(1,1). Select the **Matrix** button to display frequencies. When displayed by frequency, the entire matrix is presented in the Data View pane for each selected frequency. When displayed by matrix cell, the data for the individually chosen cells is shown across all frequencies. Use the **Select All** check box to select all frequencies or cells.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data View Pane

The Data View Pane displays data for **Sweep** or **Matrix** in either plots or a table, depending on your selection.



Loading Data into Network Data Explorer

You can launch **Network Data Explorer** from within several Ansys products. In the Project Manager, open <project> > **Analysis** > **Setup***n* > **Sweep***n*. Right click on the sweep and click **Network Data Explorer** in the shortcut menu. The current solution data is automatically loaded and ready for viewing. Otherwise, you must load a data file into Network Data Explorer.

> **Note:** When solution data that is loaded into NDE is modified and resimulated in another Ansys product, the NDE data automatically updates.

You can import the following file types:

- Touchstone Format (*.s*p)
- Touchstone 2 Format (*.ts, *.sp)
- Citifile (*.cit)
- Neutral Format (*.nmf)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- State Space File (*.sss)

> **Note:**
>
> When this type of file is loaded, Network Data Explorer regenerates s-parameter data based on the file.
>
> You can compare the regenerated s-parameters to the original data.

To import a file into Network Data Explorer, either drag and drop an analysis from the Project Manager into Network Data Explorer or

1. On the **NDE** ribbon, click **Open**. An **Open** window opens.
2. Navigate to and select a file.
3. Click **Open**. The file appears in the **Files** tree.

The file browser allows you to open multiple files at a time. However, the displayed data always corresponds to the data set indicated in the Network Data Selection pane. Click the file you want in that pane to switch between data sets.

Exporting Data from Network Data Explorer

Network Data Explorer allows you to export data to a variety of different file formats.

In this section, you will learn about:

Exporting SYZ Data

Exporting Macro Model

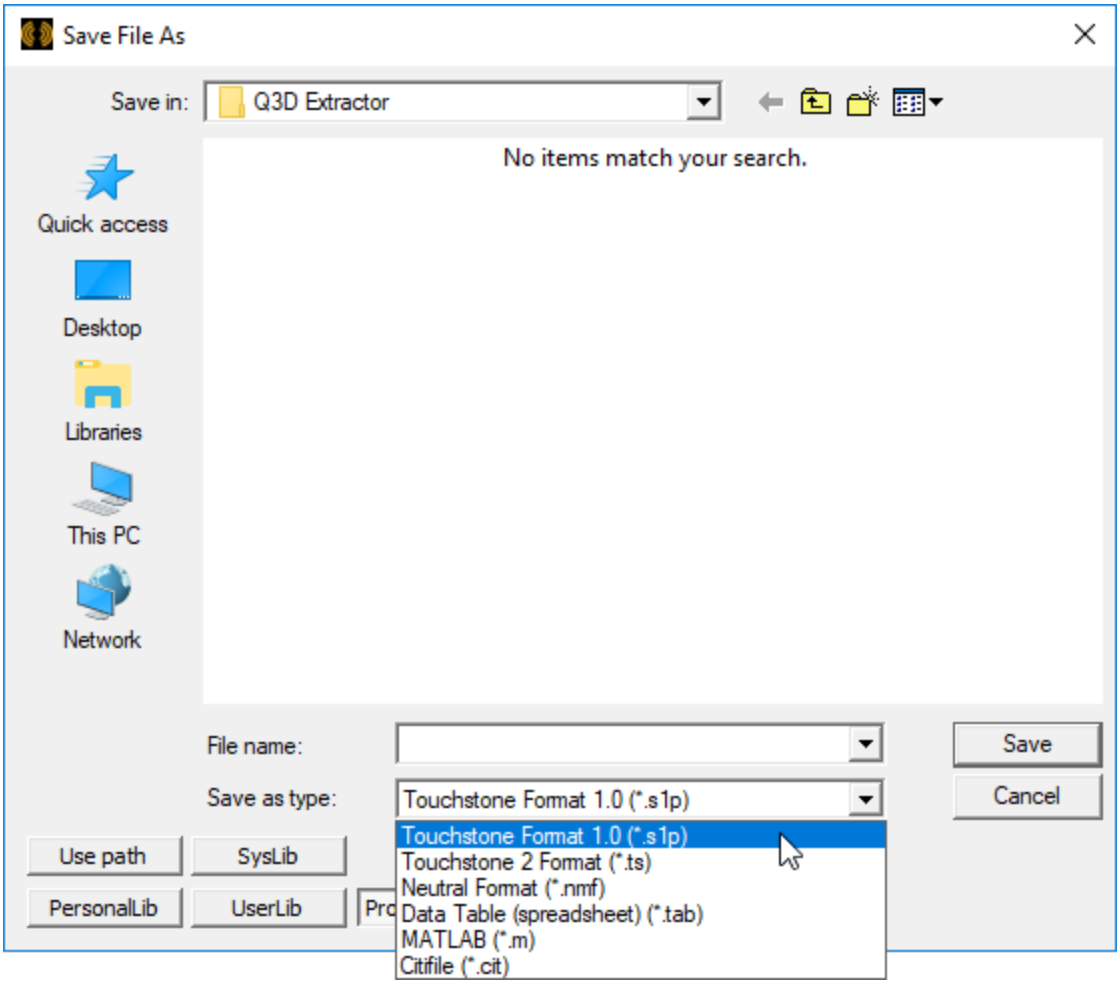
Creating an NPort Model

Scripting for Network Data Explorer

Exporting SYZ Data

To export SYZ data from within Network Data Explorer, click the **Save As** icon on the **NDE** ribbon. The **Save File As** 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.

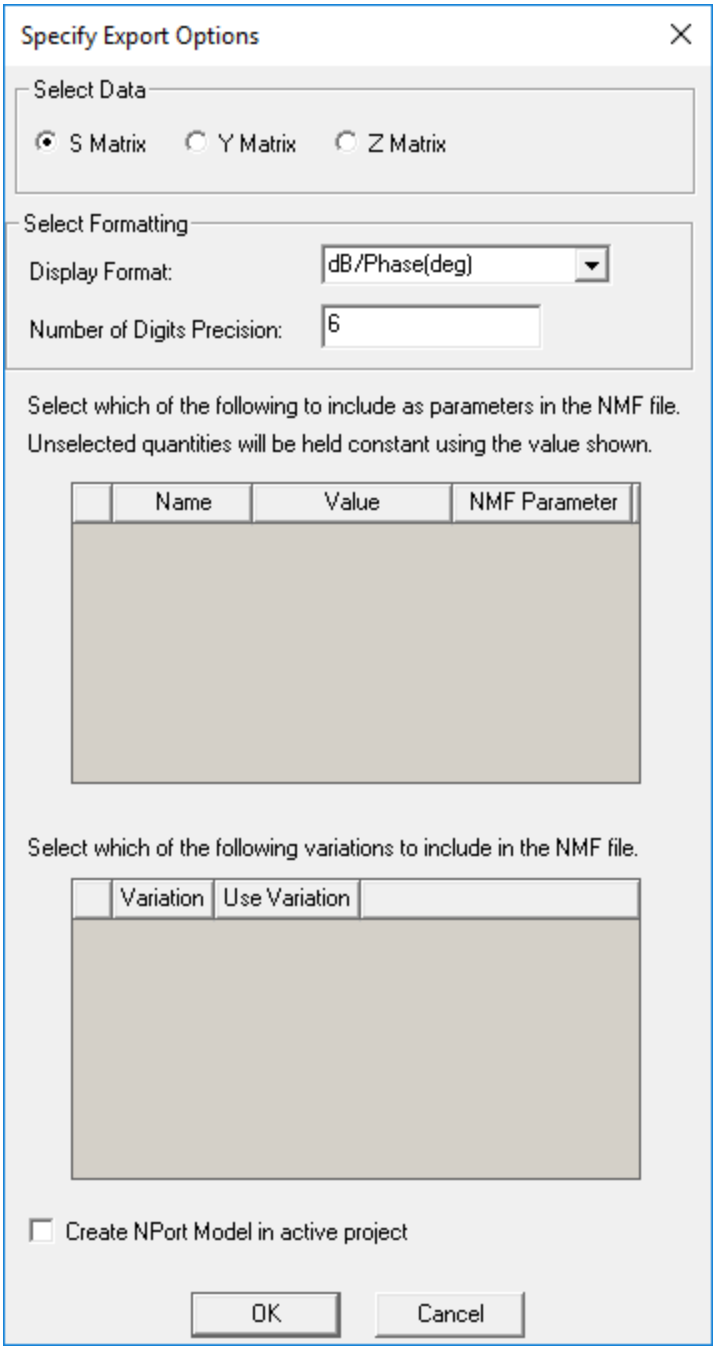


You can export data in any of six file types:

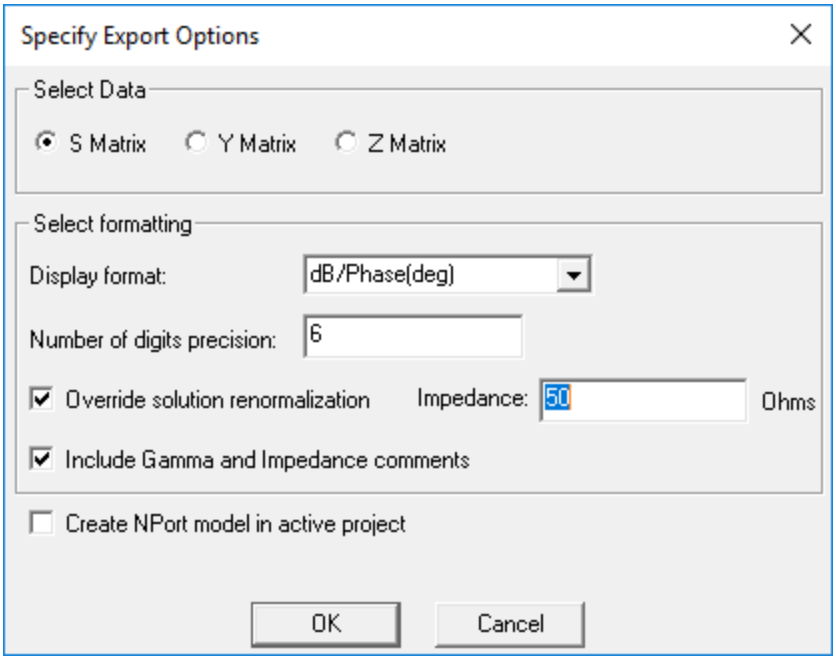
- Touchstone Format 1.0 (*.s*p)
- Touchstone 2 Format (*.ts)
- Neutral Format (*.nmf)
- Data Table Spreadsheet (*.tab)
- MATLAB (*.m)
- Citifile (*.cit)

Select a file type and name for export. A **Specify Export 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.



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 type of export file, different options appear. However, all file types allow you to:

- Select from **S Matrix**, **Y Matrix**, and **Z Matrix** data.
- Select the **Display Format**.

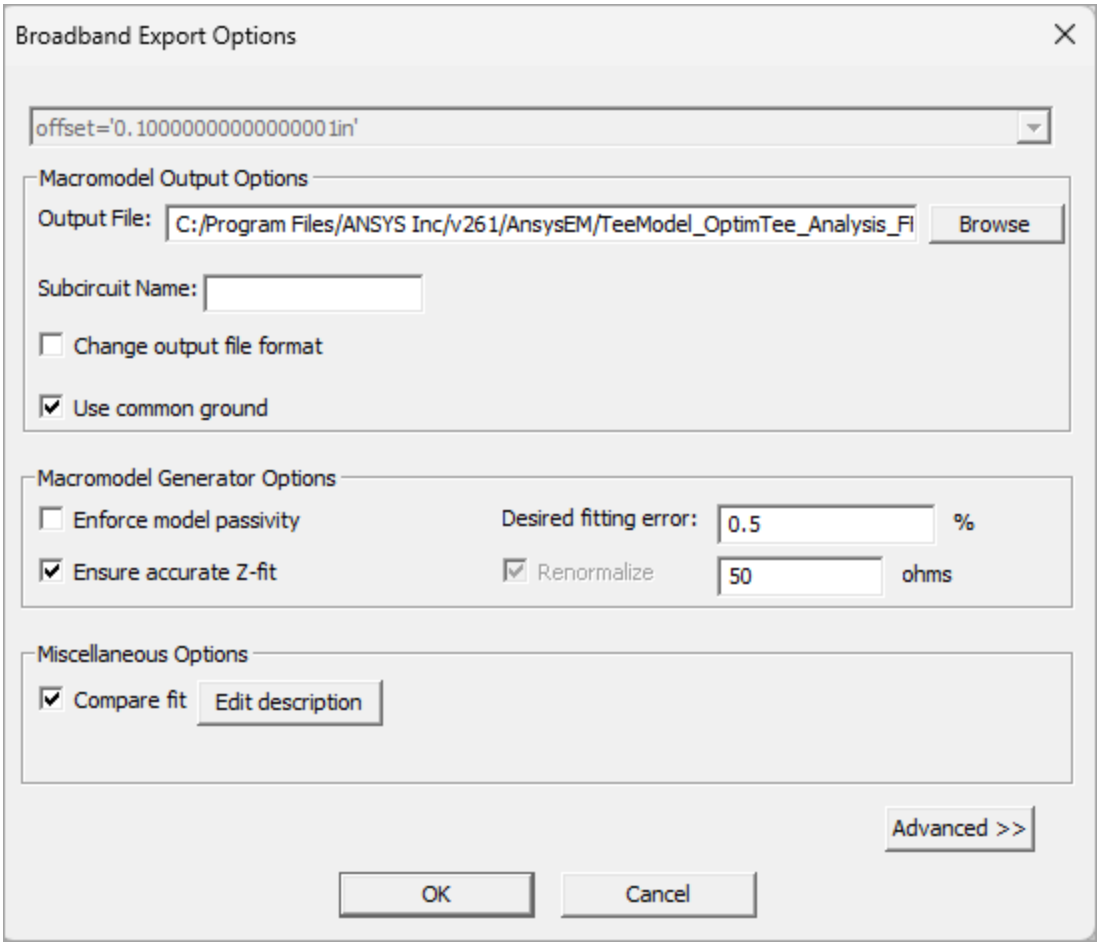
Some types allow you to create an NPort Model in the active project.

Exporting Macromodel

Network Data Explorer lets you export macro model data. To export data, click the **Broadband** icon on the **NDE** ribbon.

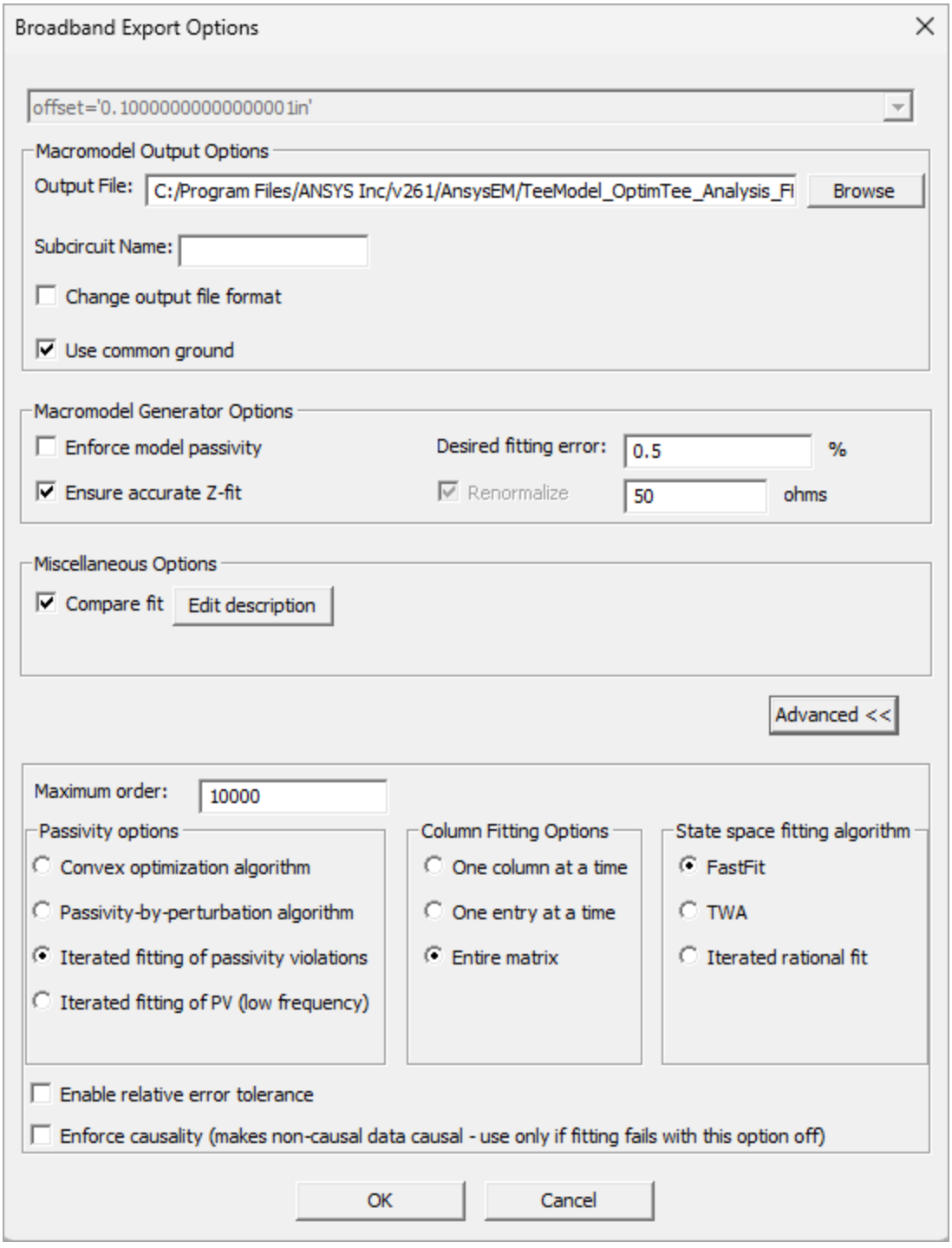
The **Broadband Export Options** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **Advanced >>** to view all options.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Macromodel Output Options include:

- **Output File** – Allows you to choose the name and location of the file.
- **Subcircuit Name** – Use this field to name the subcircuit.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Change Output File Format** – Check this box to open a submenu allowing you to select a new output format.
- **Use Common Ground** – Check this box to use common ground. When this option is on, ports are referenced to ideal ground (node 0). When this option is off, extra ports are generated to provide the reference levels. Common grounding is best when the pins are physically near to each other and ideal ground is suitable. For distant connections and circuits with non-ideal reference levels such as differential pairs, common grounding is not used.

> **Note:**
>
> - R and L values may be quite sensitive to the values of the S-parameters. This is an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
> - Because resistances of power cables is typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to get multiplied by 5000 times when the R and L values are computed.
> - As a general rule, for high power applications a reference impedance of 1 ohm is probably a better choice than 50 ohms.

Macromodel Generator Options include

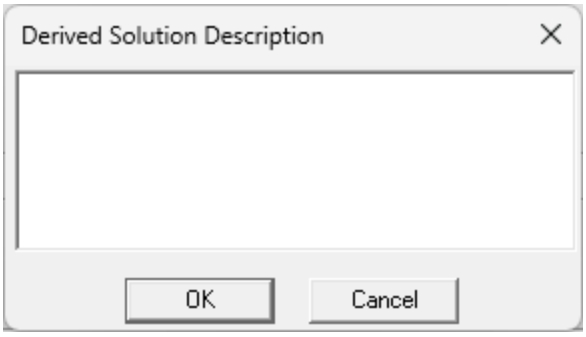
- **Enforce Model Passivity** – Check this box to enforce passivity.
- **Ensure Accurate Z-fit** – Check this box when state-space fitting of Y-parameters or Z-parameters does not produce an accurate fit.
- **Desired Fitting Error** – Allows you to select the value at which rational fitting fails (if the fitting error exceeds this value).
- **Renormalize** – Check this box to renormalize using the specified impedance item. 50 ohms is the default setting, but you can type a different value.

> **Note:**
>
> - R and L values may be sensitive to S-parameter values. This presents an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
> - Since resistances of power cables are typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to be multiplied by 5000 when the R and L values are computed.
> - For high-power applications, a reference impedance of 1 ohm is generally a better choice than 50 ohms.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Miscellaneous Options include

- **Compare Fit** – When this box is checked, the original and derived solution will be available for comparison. You can click **Edit Description** to open the **Derived Solution Description** window and add a text description to better identify the export.



Advanced Options include:

- **Maximum Order** – Allows you to specify the number of poles. See Note below.
- **Passivity Options** – If you enabled **Enforce Model Passivity**, this area allows you to select the passivity enforcement method.
  - **Convex optimization algorithm** – guarantees a passive state-space realization, but is very slow and memory-intensive. Not practical for numbers of ports beyond ten.
  - **Passivity-by-perturbation algorithm** – designed for systems with a large number of ports. Less accurate than the Convex optimization method.
  - **Iterated fitting of passivity violations (IFPV)** (default) – less accurate than other algorithms but more suitable for larger numbers of ports.
  - **Iterated fitting of PV (low frequency)** – similar to IFPV while improving the fit to "Z" at DC and low frequencies. A better choice of passivity enforcement when the fit to corresponding "Z-data" is important such as power delivery, EMI/EMC applications.

    > **Note:**
    >
    > For a more detailed explanation on any of the passivity options, see the technical note section of the Circuit help.

- **Column Fitting Options** – This area allows you to choose how poles are matched to columns:
  - **One Column at a Time** – The set of poles will be shared across all entries of a single column.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **One Entry at a Time** – Each entry will be fitted using a separate set of poles.
- **Entire Matrix** – The set of poles will be shared across all entries of the matrix being fitted.

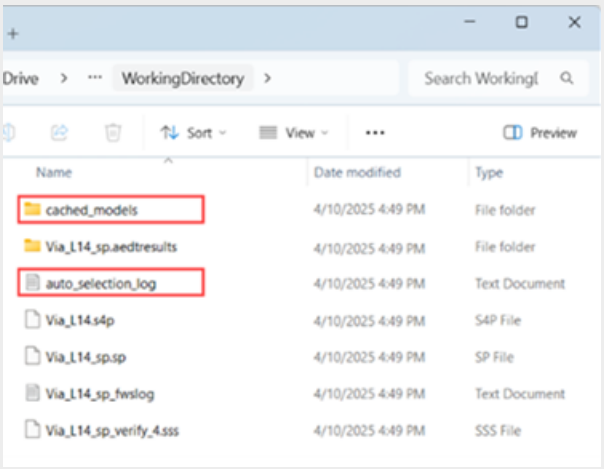
> **Note:**
>
> - Typically, using the same set for all entries is adequate, and yields the most compact models. However, if all the entries of the matrix have completely unrelated transfer functions, it may be better to fit them using separate pole sets.
> - The options **One column at a time** and **One entry at a time** do not work when either **Ensure accurate *Z*-fit** or FastFit is used.

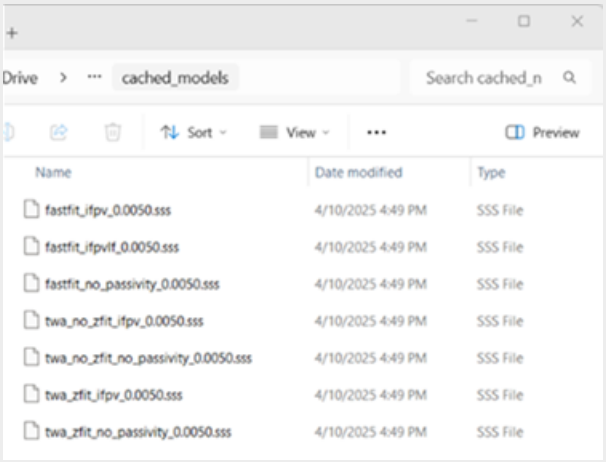
- **State Space Fitting Algorithm** – Allows you to select FastFit, TWA, Iterated rational fitting, Auto Fast, or Auto Accurate.
  - **FastFit** (default) – FastFit is the Ansys-proprietary method for state-space fitting. Network Data Explorer uses FastFit for calculating the state-space matrices from the network data. The FastFit algorithm for state-space fitting is an alternative to the Tsuk-White algorithm (TWA) and Iterated Rational Fitting (IRF) methods. FastFit is generally as accurate as TWA, but is significantly faster than both TWA and IRF. It also aims to fit the lower frequencies with higher fidelity.
  - **TWA** – The Tsuk-White Algorithm is an Ansys-proprietary method for fitting a state space model to extracted s-parameter data. It uses techniques based on Singular Value Decomposition (SVD) to quickly determine required number of poles for fitting a model.
  - **Iterated Rational Function** – The IRF fitting approach takes a matrix of S-parameter data and, for each matrix entry, tries a succession of different pole-zero approximations (increasing the number of poles used at each iteration) until it can find an acceptable fit to the data. For broad frequency sweeps and large numbers of excitations, this process can be time consuming because of all the iterations and is not guaranteed to produce a good fit to the data. It is retained as a fallback if the TWA algorithm fails.
  - **Auto Fast** – (Beta) To use the Auto Fast option, first enable **State Space Fitting Auto Selection** in the Beta Options dialog. The Auto Fast option runs all the fitting options in parallel and returns the first model that meets the user's specifications. It takes less time, however, the model may not be the one with the least fitting error.
  - **Auto Accurate** – (Beta) To use the Auto Accurate option, first enable **State Space Fitting Auto Selection** in the Beta Options dialog. The Auto Accurate option runs all the fitting options to completion, then returns the model with the least fitting error. This option may take longer but it is guaranteed to return the best fit model.

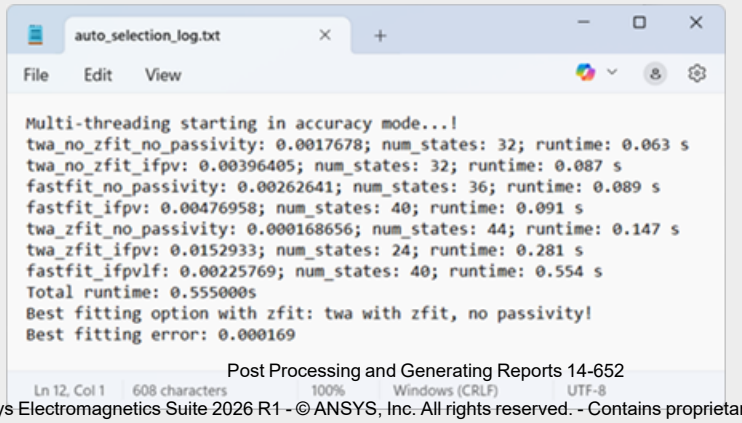
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

After a successful completion of Auto Fast or Auto Accurate, a new folder and a log file (.txt) is created in the working directory. The folder, named cached_models, contains the .sss files from the parallel runs of the different state-space fitting options. The log file, named auto_selection_log.txt, contains the status of each fitting thread.







Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

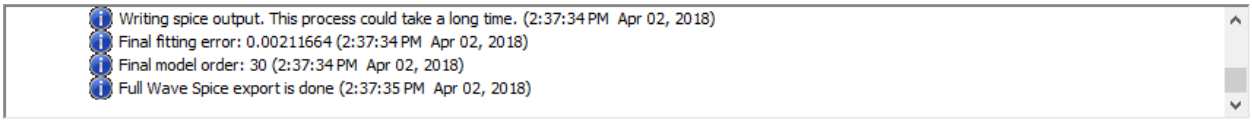
- **Enable Relative Error Tolerance** – Allows you to enable relative error tolerance, which works best with TWA fitting.

  > **Note:**
  >
  > The **Enable Relative Error Tolerance** option works best with the TWA fitting algorithm, is not recommended for use with iterated rational fitting, and is disabled when either FastFit or **Ensure accurate Z-fit** is used.

- **Enforce Causality** – Allows you to make non-causal data causal. Use this option only if fitting fails without it.

  > **Note:**
  >
  > Broadband models are built from a rational-function approximation of the data. The fidelity of this approximation can be controlled by setting the Maximum order (number of poles).

Click **OK** to begin the export. The **Messages** pane details the export process.



Comparing Original S-Parameters with Exported S-Parameters

If **Compare Fit** was checked during export, the **Data Selection** pane updates to list both the original and exported solution *and* the **Compare** checkbox is checked.

Creating an NPort Model

Touchstone, Neutral Format, and Citifile exports allow you to select an option to **Create NPort Model in active project**.

Creating an NPort Model exports the active data set back to the design, either as a static n-port model or as a parametric n-port model. If there are multiple variations, a parametric n-port model is automatically created containing all variations. When only a single variation exists, the user can create a static n-port model that either links to a file (the data is exported first) or stores the data itself. Linking to a file reduces the size of the ADSN file. The newly created model can be placed as a component in a circuit. This is particularly useful after reducing the number of ports via termination.

The import of the exported solution is done by reading the exported Touchstone file.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scripting for Network Data Explorer

Scripting is available for each Network Data Explorer export method, which means a script can be recorded to duplicate the export process.

Network Data Explorer can be invoked in the following contexts, and scripting is available from the Project Context and the Design Instance Context (simulation setup):

- **Project Context** – In the context of a project, you can open a touchstone file and then export.
- **Design Instance Context** – In the context of a solution (RCM from the simulation setup in a design), you can export the corresponding network data solution.

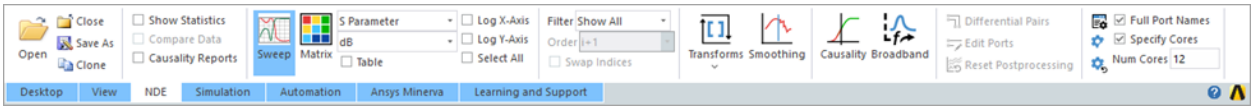
When there is no design available, export functionality and scripting are not available.

When there is no design available, export functionality and scripting are not available.

For more information, see Network Data Explorer Script Commands in the *Ansys Electronics Desktop Scripting Guide*.

Network Data Explorer Ribbon

The **NDE** ribbon provides access to many of the Network Data Explorer's functions.



In this section, you will learn about:

Data Sources

Setting Display Format

Displaying Full Port Names

Saving or Resetting Default Settings

Smoothing All Frequencies

Cell Filtering

Editing Port Properties

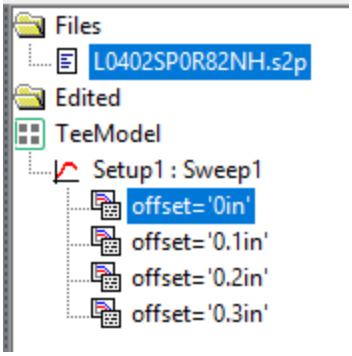
Defining Differential Pairs and Displaying Mixed Mode Parameters

Resetting All Port Properties

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

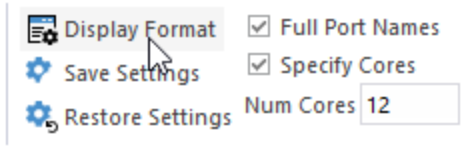
Network Data Explorer Data Sources

Network Data Explorer allows you to easily view all data sources in the **Network Data Selection** pane.

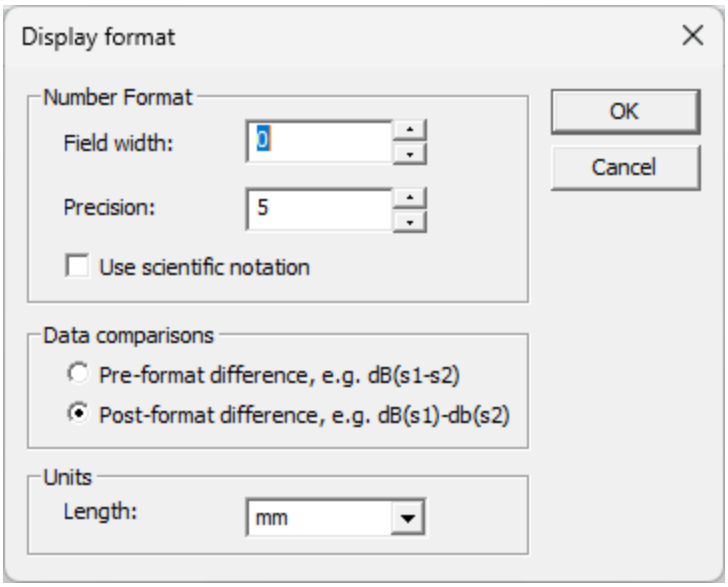


Network Data Explorer Display Format

The **Display Format** window affords additional control over the display of values in Network Data Explorer. On the **NDE** ribbon, click the **Display Format** icon.



The **Display Format** window appears.



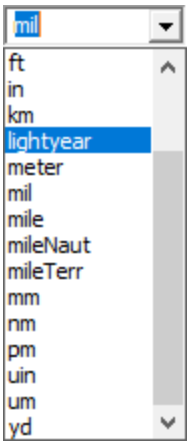
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Number Format** options allow you to specify **Field Width** (the minimum number of characters used to display a number) and **Precision** (the number of decimals to display). You can also check the **Use scientific notation** check box, if desired.

The **Data comparisons** options allow you to choose Post-format difference or Pre-format difference.

- **Pre-format difference** – when comparing data sets, subtract values before applying the formatting function (e.g. dB, magnitude); the values displayed will be the magnitude, dB, etc., of the complex difference.
- **Post-format difference** – when comparing data sets, subtract values after applying the formatting function (e.g. dB, magnitude); the values displayed will be the difference between the magnitude, dB, and so on.

The **Units** option allows you to specify the **Length** unit (the unit used to display and interpret length values). The default is mm.

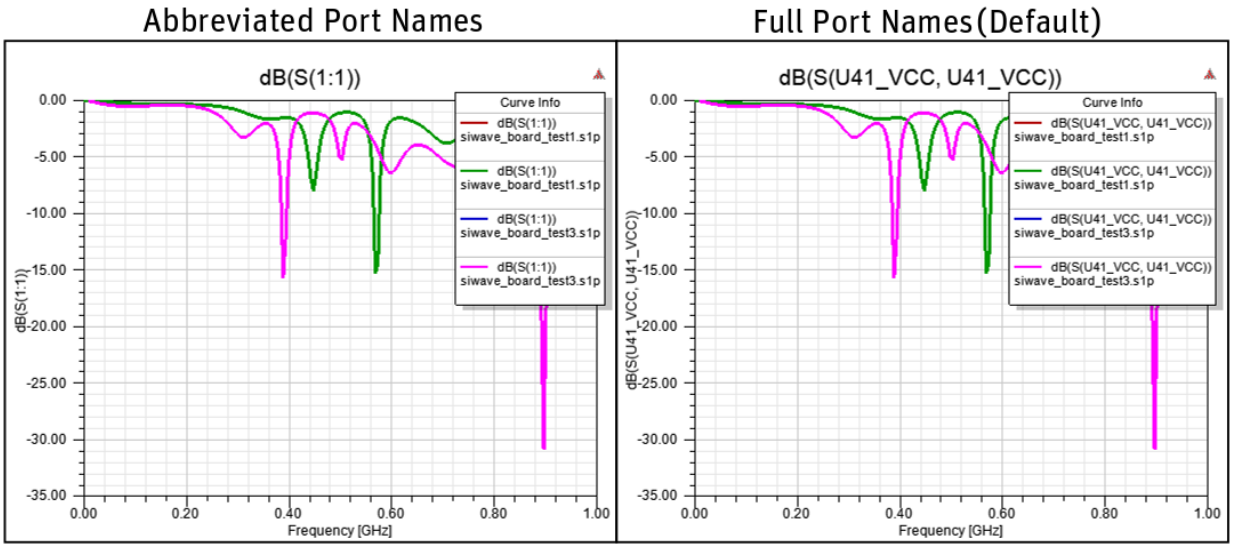


Network Data Explorer Display Full Port Names

By default, full port names are displayed. This applies to both the Data Selection pane and the Data View pane. To change this so that port names in Network Data Explorer are displayed in an abbreviated form (P1, P2, etc.), click the **Full Port Names** check box on the **NDE** ribbon.

The following figure shows the difference in display for a plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Tool-tips always display the full port name.

Network Data Explorer Save or Reset Default Settings

To save field settings as the default, click the **Save Settings** icon on the **NDE** ribbon. The next time Network Data Explorer is opened, the chosen settings will be selected by default.

To restore the default settings (i.e. the settings previously saved using the **Save Settings** button), click the **Restore Settings** icon on the **NDE** ribbon. The next time Network Data Explorer is opened, the original settings will be selected by default.

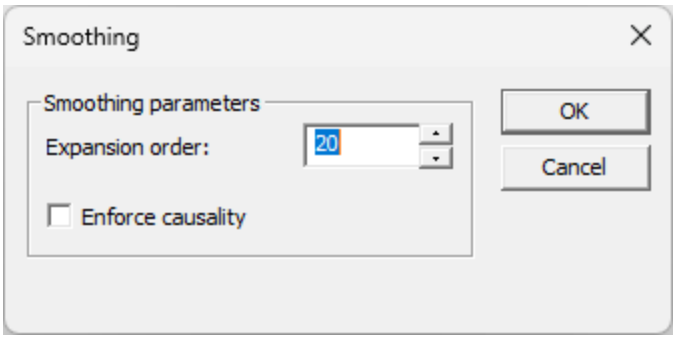
Smoothing

To access data smoothing options:

1. Click the **Smoothing** icon on the **NDE** ribbon.

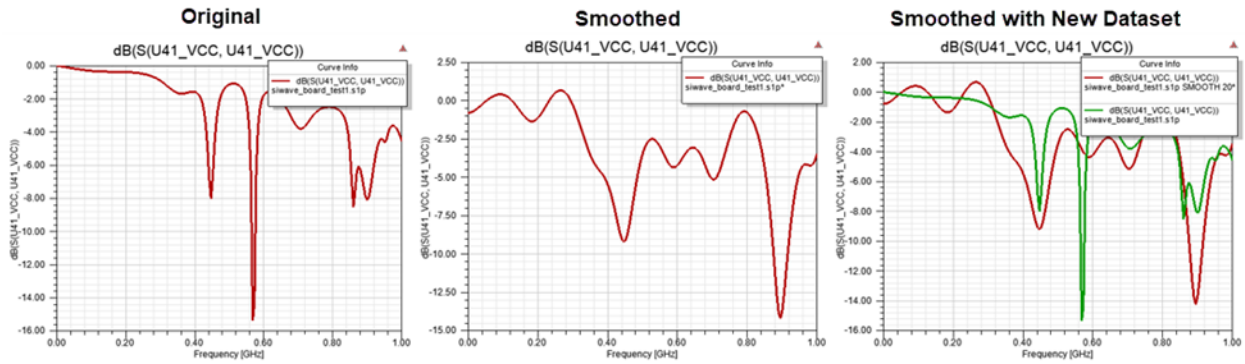
The **Smoothing** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. The **Smoothing Parameters** area allows you to choose the **Expansion Order**. This can be any discrete value between 1 and 150.
3. If desired, check the **Enforce Causality** check box.
4. If desired, check the **Create a new data set containing the smoothed data** check box. If selected, the smoothed data appears alongside the original data.
5. Click **OK**.

The Data View Pane updates. The following image shows a plotted result with **Create a new data set containing the smoothed data** unchecked and checked.

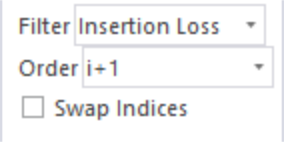


A least-squares polynomial fit of the specified order is used to interpolate new data points for the magnitude and phase components of the S-parameters.

Cell Filtering

The cells available in the data selection pane may be restricted using cell filtering. The **Cell Filtering** control are located on the **NDE** ribbon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cell filtering is modeless, and filters are immediately applied to the cell list. Filtering remains in effect when the window has been closed.

For an n-port model with a total of n pins, the choices in the **Filter** drop down menu are:

- **Show all** – display all available cells. There are n2 lines.
- **Return loss** – show diagonal matrix elements S(i,i). There are n lines.
- **Insertion loss** – see description of **Order** drop down menu options below. With half the pins on the left side of the symbol and half on the right, it shows S(left, right). There are n/2 lines.
- **Lower Triangle** – shows all matrix elements below the diagonal. There are (n2 – n) / 2 lines.
- **Upper Triangle** – shows all matrix elements above the diagonal. There are (n2 – n) / 2 lines.

The **Order** drop down menu is only enabled for Insertion loss and has three options. They describe the arrangement of pins on two sides of a symbol. Half the pins are on the left side and half on the right side. The insertion loss is shown between a left pin and a right pin that are opposite each other. As an example, the first line of a 10-port matrix is given for each option description below.

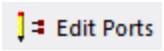
- **i + 1** – The pins are placed in the symbol by alternating sides, so left side has odd index entries and right side has even index entries. Example: S(1,2)
- **i + n** – The pins are placed on the symbol in order, the first half on the left and the second half on the right. Example: S(1,6)
- **2*n – i +1** – The pins are placed on the symbol, the first half down the left side, the second half on the right side, but the right side is in reverse order. Example: S(1,10)

The **Swap Indices** check box is only enabled for Insertion loss. When checked, it will show the matrix entry with the indices swapped. Example: S(1,6) becomes S(6,1)

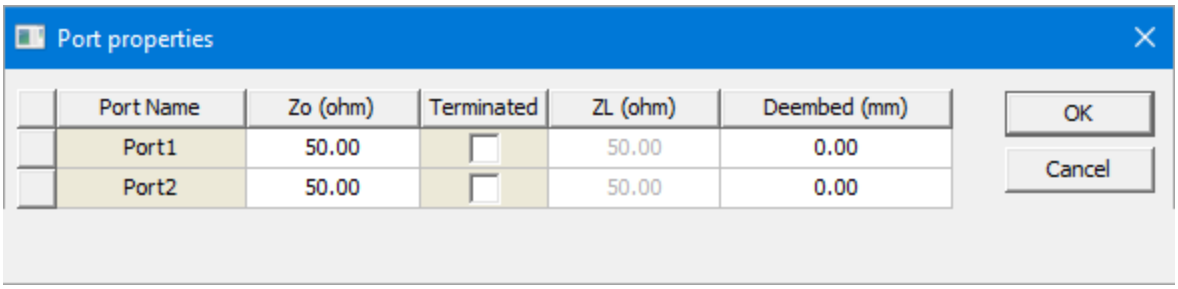
Changing Port Properties and Reducing Matrix Size

The normalization impedance, termination, port order, gamma values, and deembedding distance may all be edited from the **Port 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 access these options, click **Edit Ports** 

on the **NDE** tab. The **Ports Properties** window opens.



> **Note:** For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution.

Ports appear in a table. Click a column heading to sort by that column. Click within a cell to edit the port property:

- **Zo (ohm)** and **ZL (ohm)** – specify Impedance values. Accepted syntaxes are:
  - real (e.g., 50)
  - real + imag i (e.g., 50+5i)
  - imag i (e.g., 5i).
- **Terminated** – use the check box to terminate a port. Terminated ports are eliminated from the matrix, reducing the matrix size. Existing data sets with mismatching port numbers will no longer be available for data comparisons.
- **Deembed** – this column appears only if gamma values are available. Default units can be changed from the Set Display Format window.

To reorder ports, click and drag a row to a new location.

To save changes, click **OK**.

Displaying Mixed-Mode Parameters using Differential Pairs

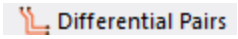
Network Data Explorer displays mixed-mode parameters when differential pairs are both defined and activated.

To define differential pairs:

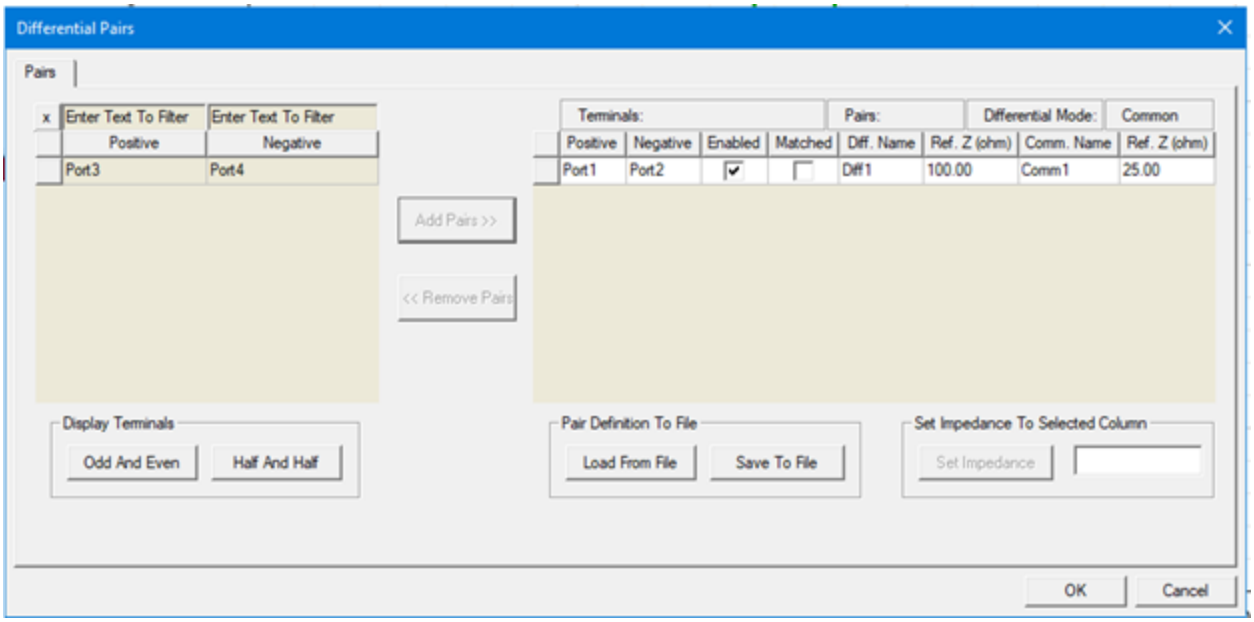
1. Select existing ports.
2. Open Network Data Explorer (**Tools**>**Network Data Explorer**).

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 **NDE** ribbon, select **Differential Pairs**.

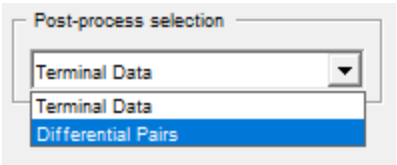


The **Differential Pairs** window appears.



4. Select a pair from the list on the left and click **Add Pairs**.
5. Click **OK**.
6. To disable all differential pairs, click the **Enabled** column header in the **Differential Pairs** dialog to deselect all pairs.

For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution. When the HFSS Driven Terminal Design has differential pairs defined, NDE shows this drop down menu and you can change between showing reports for Terminals or Differential Pairs.Select **Differential Pairs** from the **Post-process selection** field to view mixed-mode 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 Network Data Explorer **Edit** menu option **Reset All Port Properties** deactivates all pairs, but it does not clear the differential pair settings. And since **Reset All Port Properties** also clears reference impedances and terminations, it should not be used when the user simply wishes to disable all differential pairs.

Reset All Port Properties

Reset All Ports resets all changes in the **Edit Ports** window. It also deactivates all differential pairs defined in the **Differential Pairs** window but does not remove the definitions.

> **Note:** For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution.

Data View Pane Context Menus

The Data View pane presents different right-click menu options, depending on the context. Some commands are the same as those on the NDE ribbon. Others appear only in the context menus.

Network Data Explorer commands unique to the context menus are:

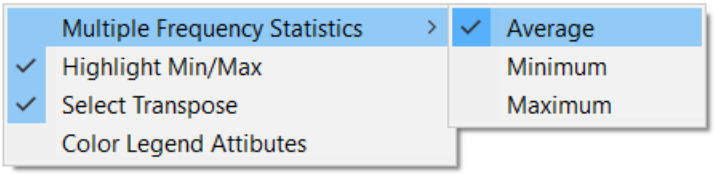
- Multiple Frequency Statistics
- Highlight Min/Max
- Select Transpose
- Color Legend Attributes
- Matrix Entries Plot Menu

Multiple Frequency Statistics

The **Multiple Frequency Statistics** menu option determines the statistical composite to display when multiple frequencies have been selected for the matrix display. The statistical data is always the first matrix displayed, followed by matrices for each individual frequency. The **Multiple Frequency Statistics** option also indicates the data used in the colored matrix plot when multiple frequencies have been selected.

This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.

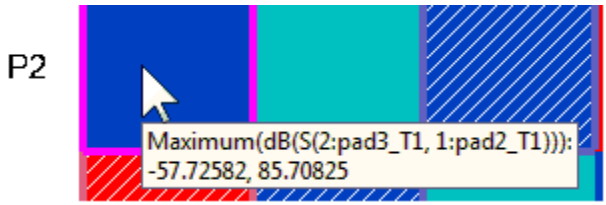
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The menu options are:

- **Average** – display the average of the matrix values across selected frequencies.
- **Minimum** – display the minimum matrix values across selected frequencies.
- **Maximum** – display the maximum matrix values across selected frequencies.

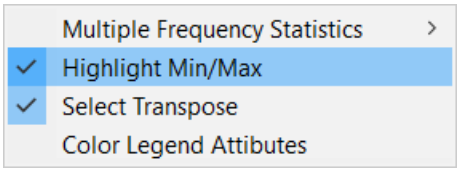
The selected information also appears in a tool-tip when you hover the cursor over a cell.



Highlight Min/Max

The **Highlight Min/Max** menu option determines whether the minimum and maximum matrix entries should be highlighted in the matrix table and color plot view.

This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.

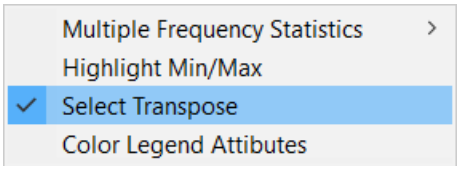


Select Transpose

The **Select Transpose** menu option determines whether transpose cells are highlighted along with selected cells.

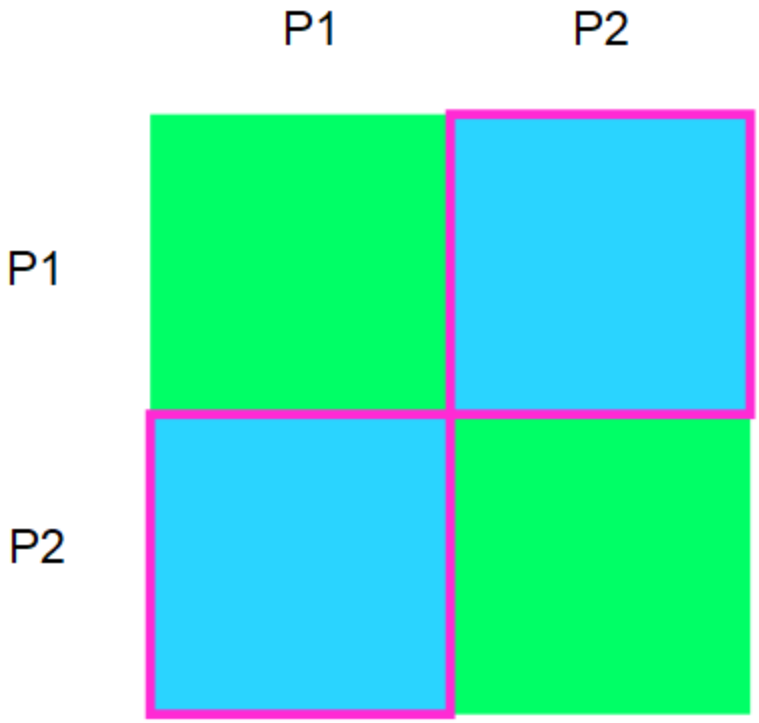
This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Transpose cells are highlighted in Table or Plot view, as shown below.

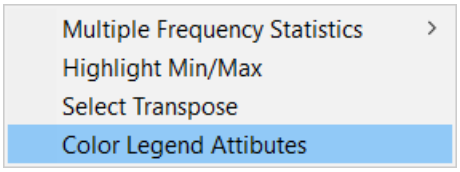
0.01000GHz	P1	-57.10340, -81.97841	-57.72581, 85.70836	-0.00213, -0.18916
	P2	-57.72582, 85.70825	-57.09556, -81.98749	-66.27113, 99.16916
	P3	-0.00213, -0.18916	-66.27110, 99.16941	-57.10246, -81.95892
	P4	-66.27112, 99.16942	-0.00213, -0.18929	-57.72556, 85.71465



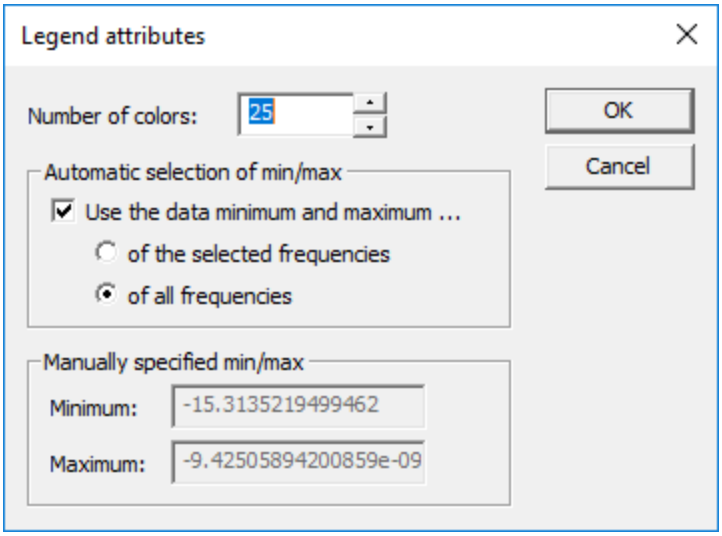
Color Legend Attributes

The **Color Legend Attributes** menu option allows you to change the granularity of the color scheme and the value range for plots. This right click menu option appears in the **Matrix** plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Alternatively, double-click the matrix plot's legend to open the **Legend Attributes** window.



Options include:

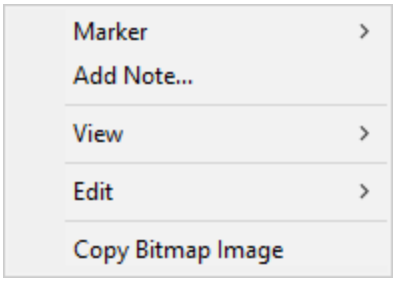
- **Number of colors** – allows you to select number of color entries in the legend (the number of divisions between the start/end of the data range). This can be set to any discrete number between 1 and 50.
- **Automatic selection of min/max** – check the **Use the data minimum and maximum** check box to automatically select the data range using the minimum and maximum values from either selected frequencies or all frequencies in the data set.
- **Manually specified min/max** – when the when the range is not automatically determined, these fields permit the user to manually enter hard values. For example, for S parameter data magnitude data, you could enter a minimum of 0 and a maximum of 1.

Using a standard range across all frequencies permits you to quantitatively compare plots, and ndExplorer remembers legend settings for each data-type and display-format pair.

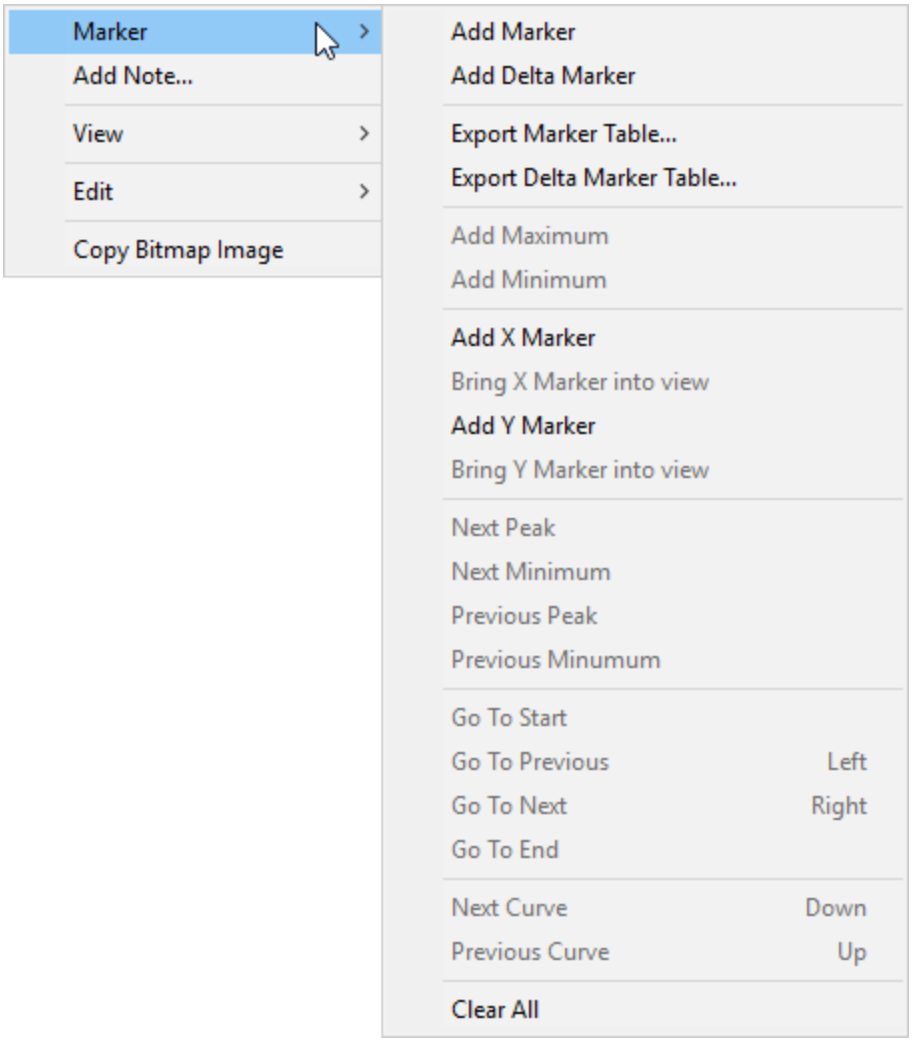
Matrix Entries Plot Menu

On the **Matrix** plot, several new right-click menu 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.

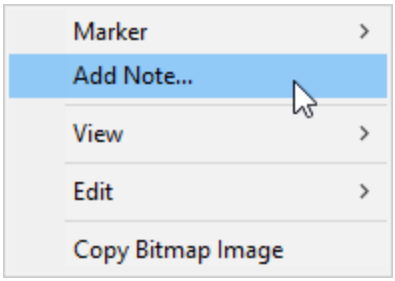


The **Marker** sub-menu provides commands for adding markers to plots.

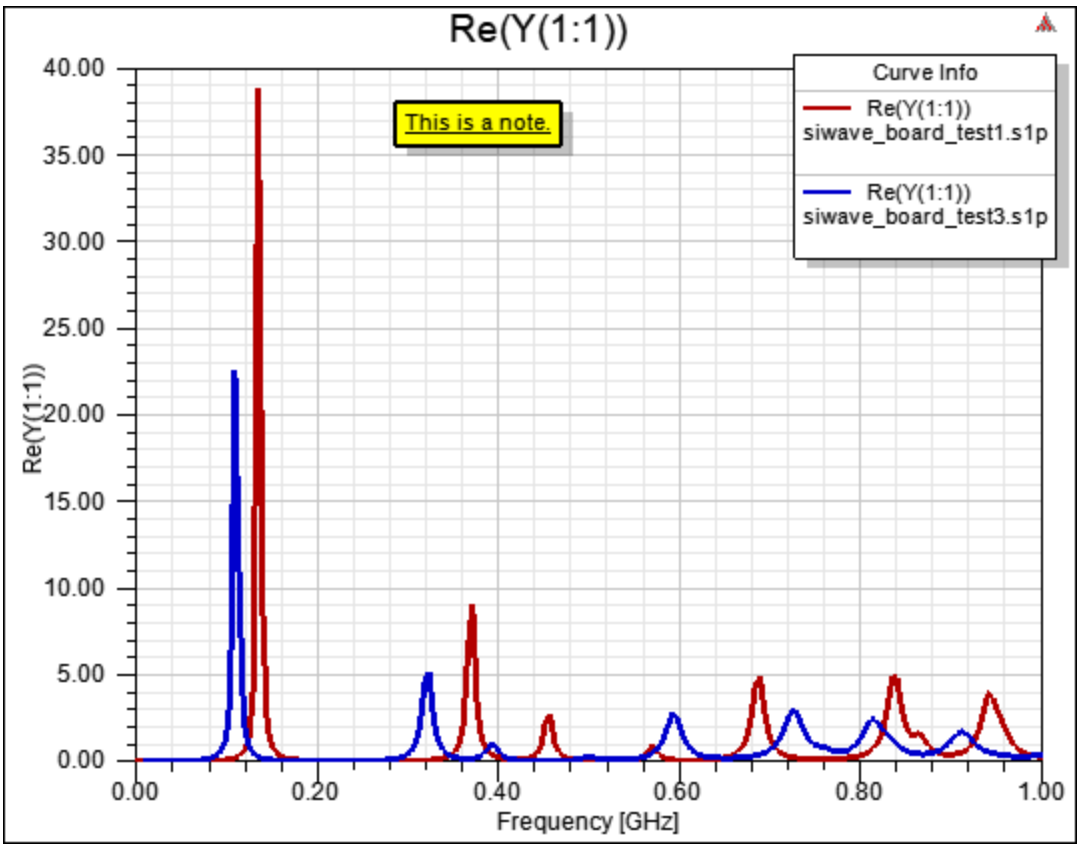


The **Add Note** menu option allows you to add a note to the plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

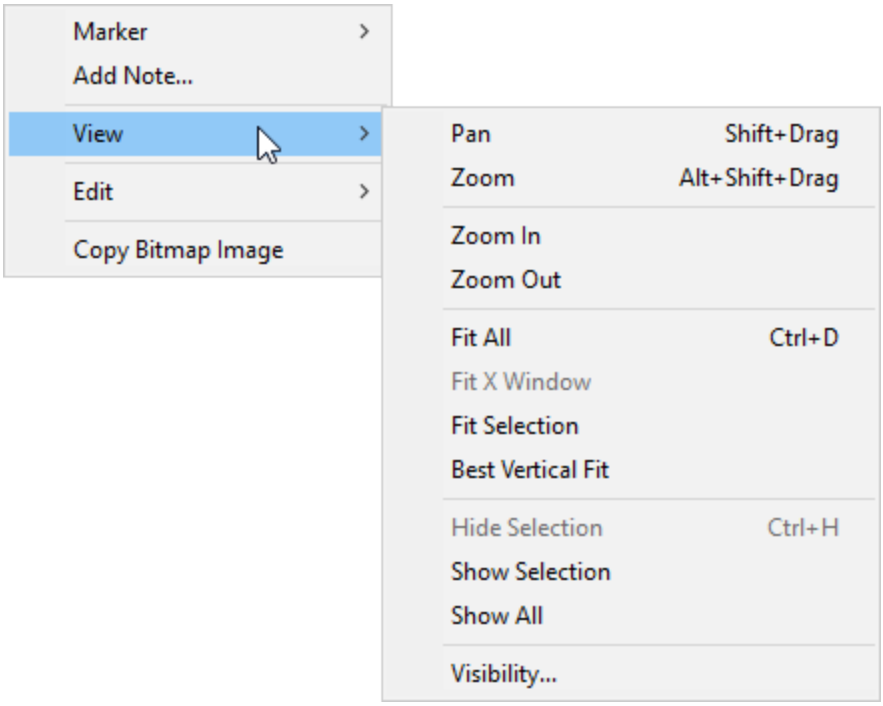


The note appears at the location you right-clicked. You can click and drag the note to a new location, or double-click the note to change its color and font.



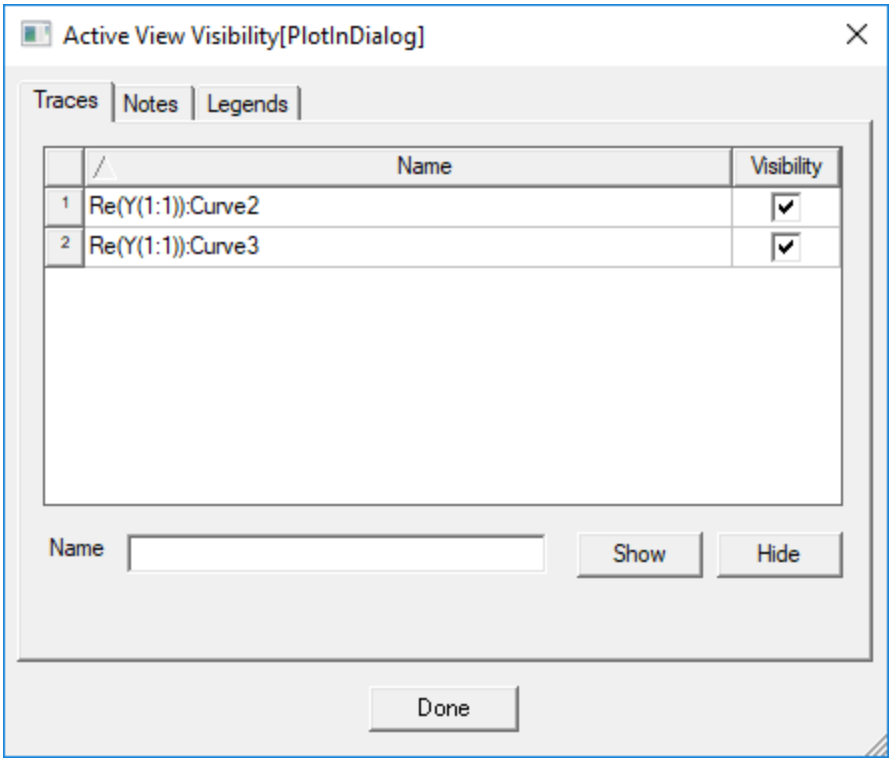
The **View** sub-menu provides commands for viewing, panning, zooming, and fitting elements to your plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

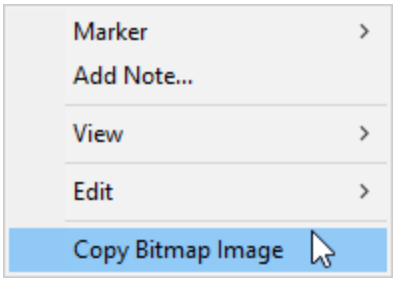


Click **Visibility** to open the **Active View Visibility** window, where you can select the visibility of traces, notes and legends.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Copy Bitmap Image** menu option copies the plot to your clipboard. You can then paste it into a graphics editor.



Exploring Network Data and Modifying the Display

Network Data Explorer allows you to view data and modify various aspects of the display, including color plots, color coding, viewing across frequencies, and displaying individual statistics. This section provides examples of Network Data Explorer capabilities:

- Viewing the S, Y, or Z Matrix for a Frequency
- Viewing a Color-coded Matrix Plot
- Displaying a Cell Graph Across All Frequencies

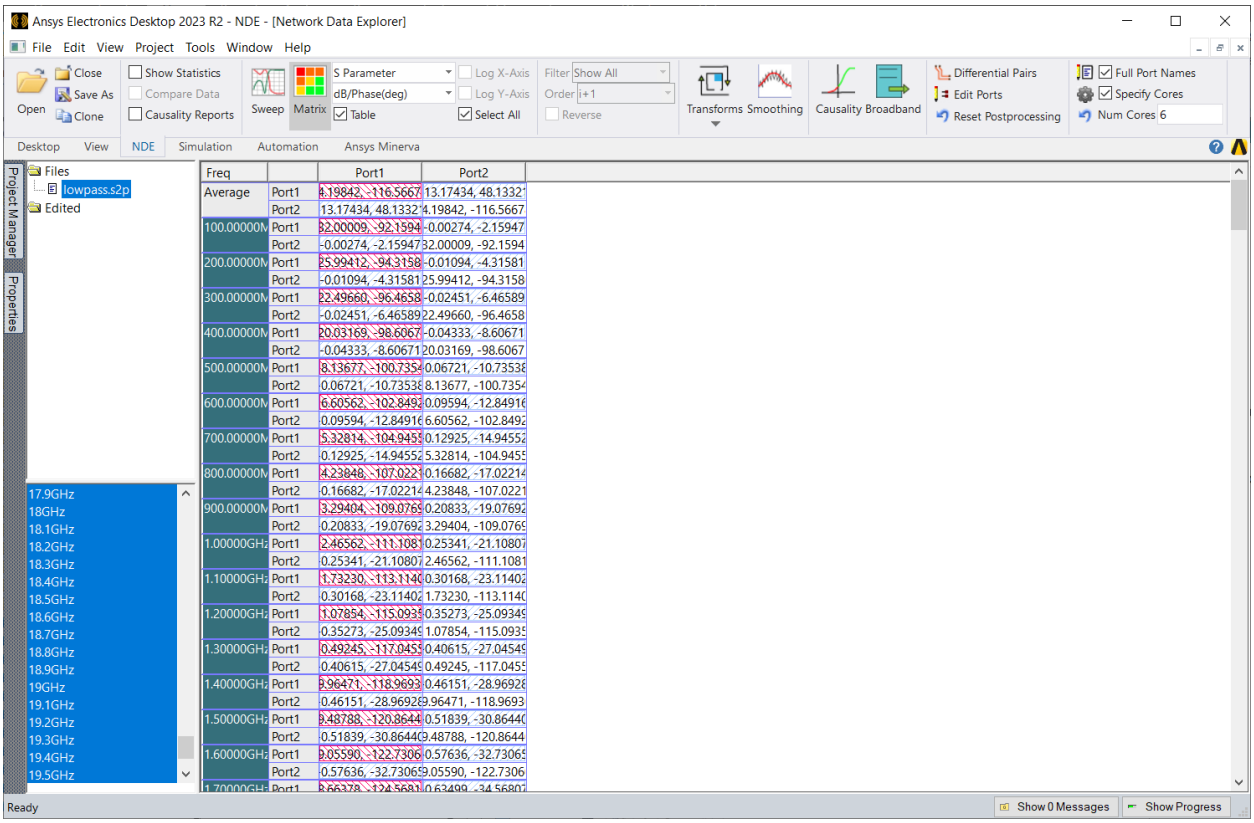
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Displaying Matrix Statistics by Frequency
- Displaying Individual Statistics for All Frequencies
- Creating a Statistics Plot
- Comparing Variations

Viewing a Matrix Table

To view the S, Y or Z matrix:

1. On the **NDE** ribbon, click **Matrix**.
2. Click the **Table** check box.
3. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.
4. Choose a format from the drop-down menu, for example dB or Mag.
5. In the Cell and Frequency Selection pane, select frequencies to display or click the **Select All** check box on the **NDE** ribbon to select all frequencies. The S, Y, or Z matrix displays.



6. Hover over a cell to see more information in a tool tip.
7. Right click to see a right click menu with other commands.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum values are highlighted in red stripes. If Select Transpose is enabled, transposes are highlighted in red stripes as well.

Minimum values are highlighted in blue stripes.

Selected cells appear in solid blue. If Select Transpose is enabled, the transpose is highlighted in blue as well.

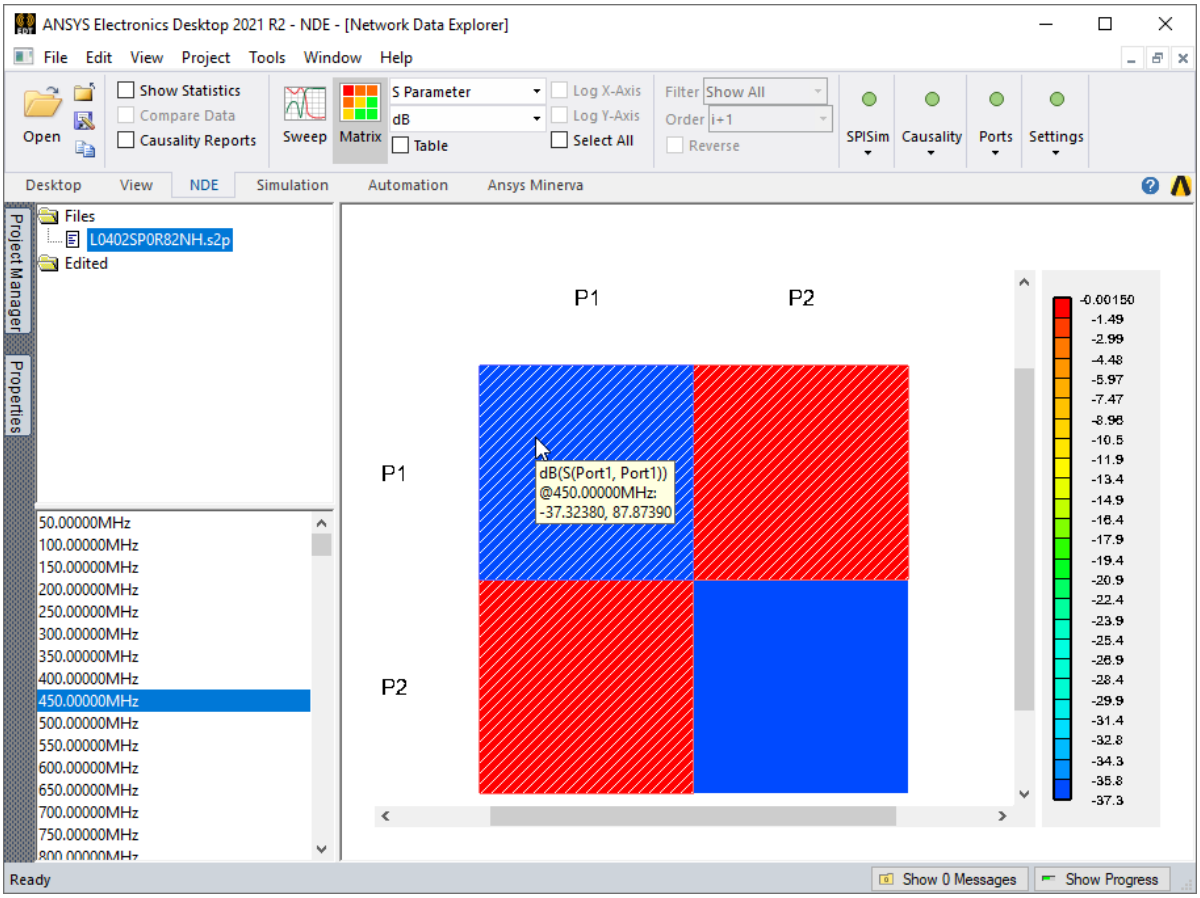
Double-clicking a cell switches to a matrix cell view, in which values for all frequencies for that cell are displayed. The double-clicked frequency is highlighted with solid red shading

Complex values are compared using their modulus. When multiple frequencies are selected, the data display depends on the Multiple Frequency Statistics setting.

Viewing a Color-Coded Matrix Plot

1. On the **NDE** ribbon, click **Matrix**.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.
3. In the Cell and Frequency Selection pane, select the frequencies you want to plot or click the **Select All** check box on the **NDE** ribbon to select all frequencies.
4. Hover over a cell to see more information in a tool tip.
5. Right click to see a right click menu with other commands.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Matrix values display in a color-coded grid. If the selected **Format** is a complex value, only the real component is used to determine the display color. When multiple frequencies or variations are selected, the data display depends on the Multiple Frequency Statistics setting. Maximum values are highlighted in red. Minimum values are highlighted in dark blue.

Hover the cursor over any cell to view information about it.

Click any cell to select it. Selected cells appear with a pink outline. If Select Transpose is enabled, the transpose is selected in pink as well.

Double-click any cell to view a matrix cell plot in which all frequency values for that matrix cell are displayed as a graph.

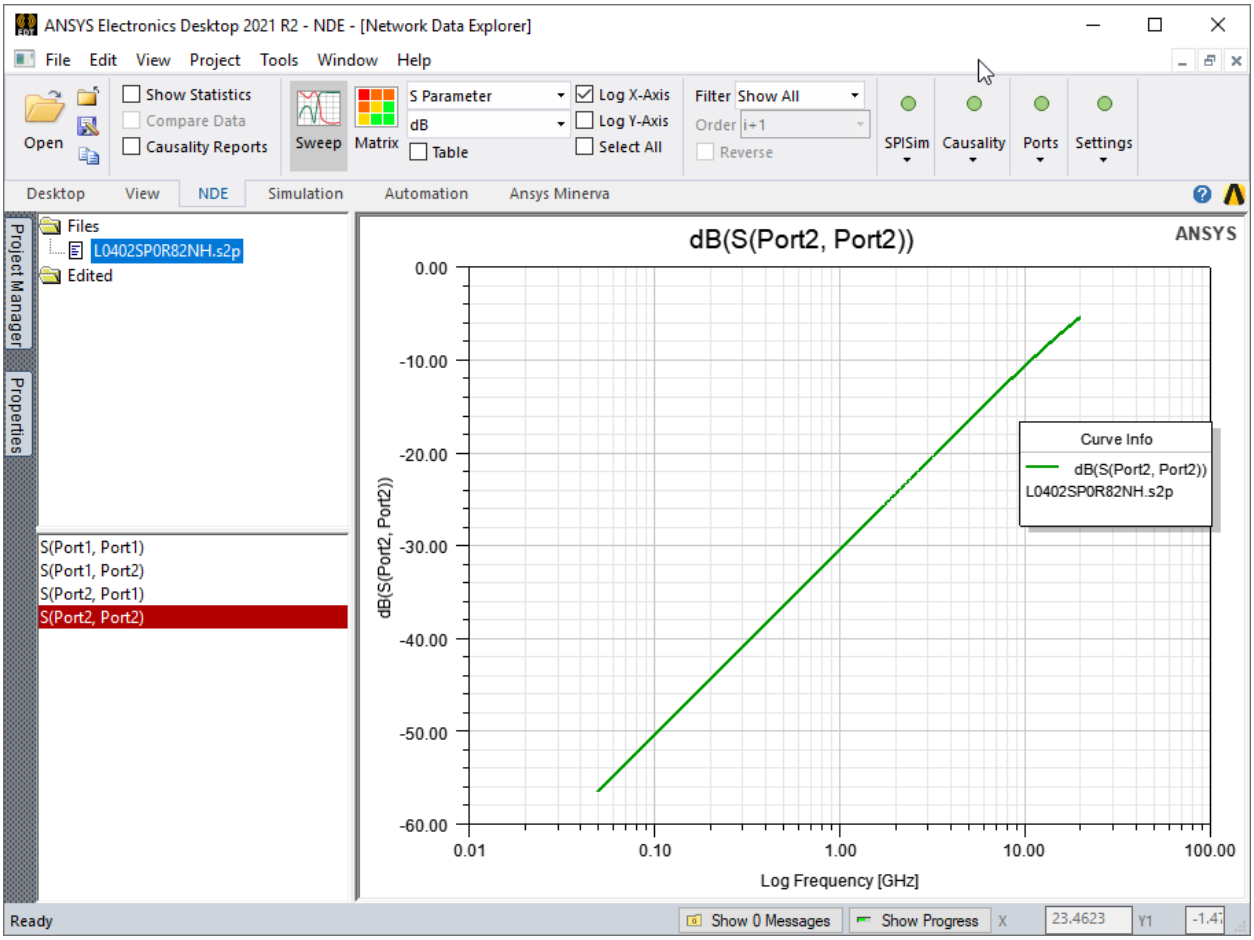
Displaying a Cell Graph Across All Frequencies

Network Data Explorer can plot a cell across all frequencies.

1. On the **NDE** ribbon, click **Sweep**. The Data View Pane updates if necessary.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, or **Z parameter**.

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 Cell and Frequency Selection pane, select cells to display or click the **Select All** check box on the **NDE** ribbon to select all cells.
4. To add a log scale to the X-axis, click **Log X-Axis** on the **NDE** ribbon.



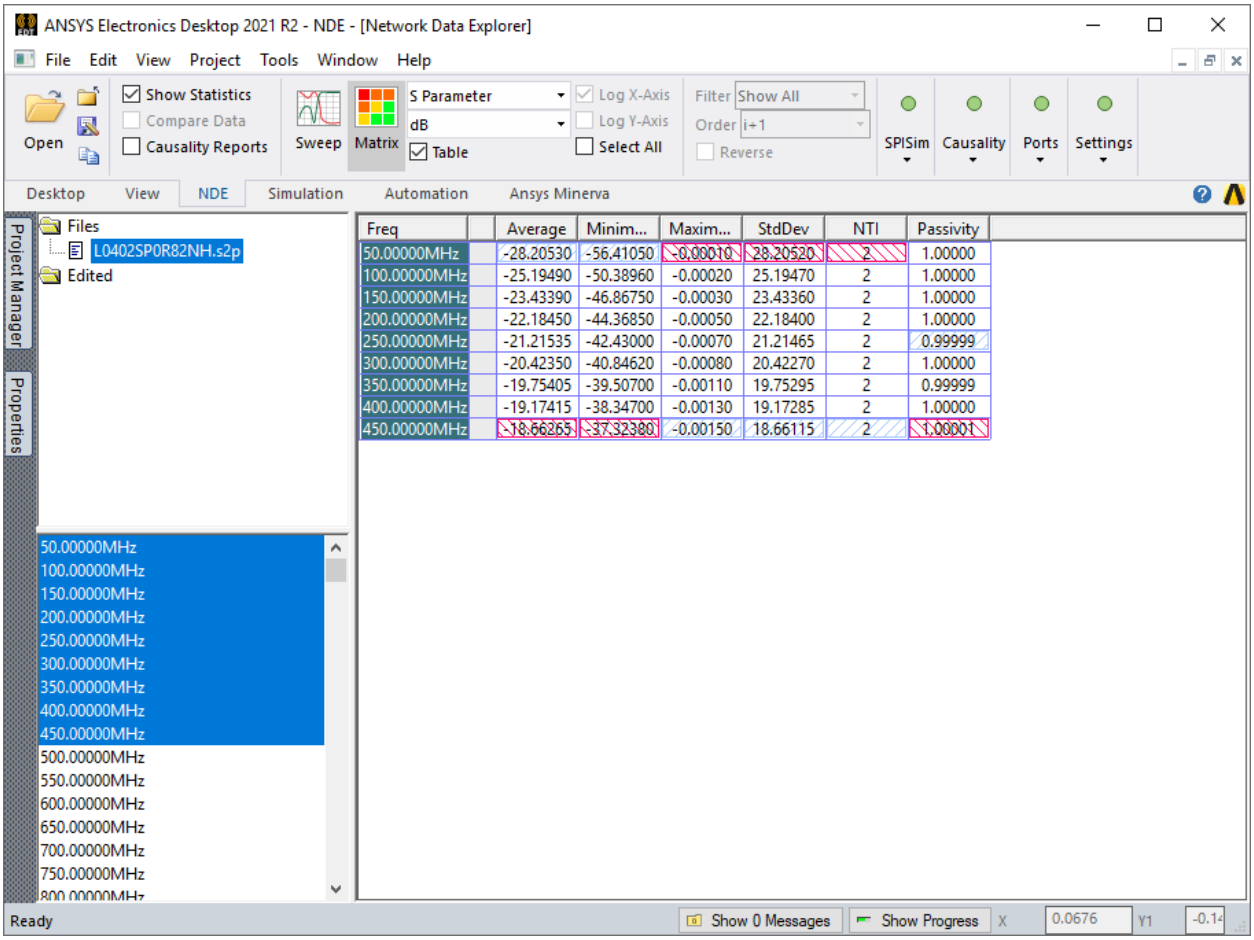
5. To add a log scale to the Y-axis, click **Log Y-Axis** on the **NDE** ribbon.
6. Right click to see a right click menu with other commands.

Displaying Matrix Statistics by Frequency

Network Data Explorer can display various statistical measurements.

1. On the **NDE** ribbon:
   a. Click **Matrix**.
   b. Click **Table**.
   c. Click **Show Statistics**.
2. In the Cell and Frequency Selection pane, select frequencies to display.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



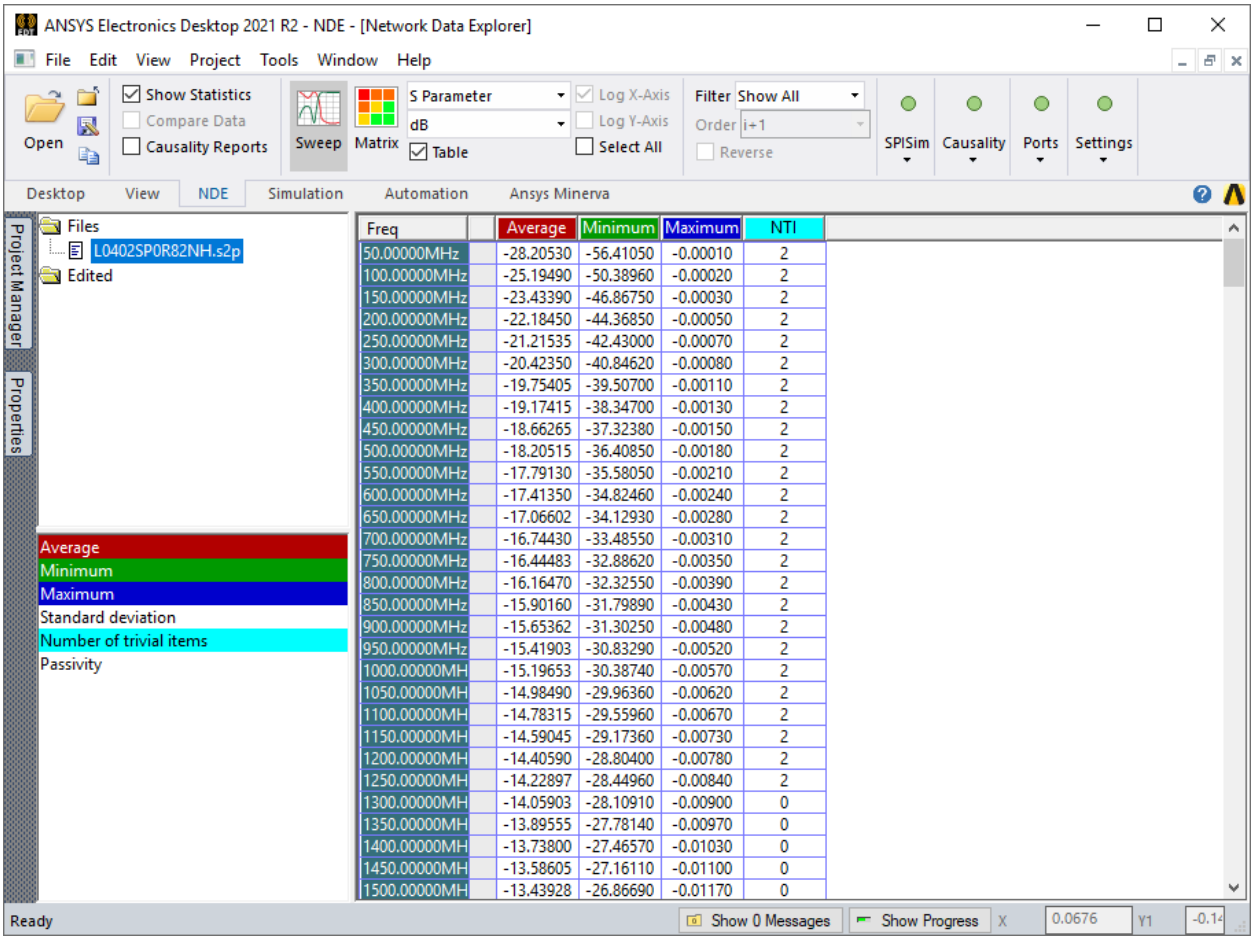
3. Click a column header to sort data by that column.
4. Hover the cursor over a cell to view information about it.

Only real (not complex) data formats are offered for statistical analysis. **Passivity** is only available for S-parameter data (comparisons inactive). **NTI** refers to the number of trivial items; for S-parameters, this includes all zeros and ones; for all other data (and data comparisons), only zeros are counted as trivial. The minimum value for each column is highlighted in blue; the maximum is highlighted in red.

Displaying Individual Statistics for All Frequencies

1. On the **NDE** ribbon:
   a. Click **Sweep**.
   b. Click **Table**.
   c. Click **Show Statistics**.
2. In the Cell and Frequency Selection pane, select statistics to display. The information displays in a table in the Data View pane.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selected statistics are displayed for all frequencies.

Passivity is only available for S-parameters (comparisons inactive).

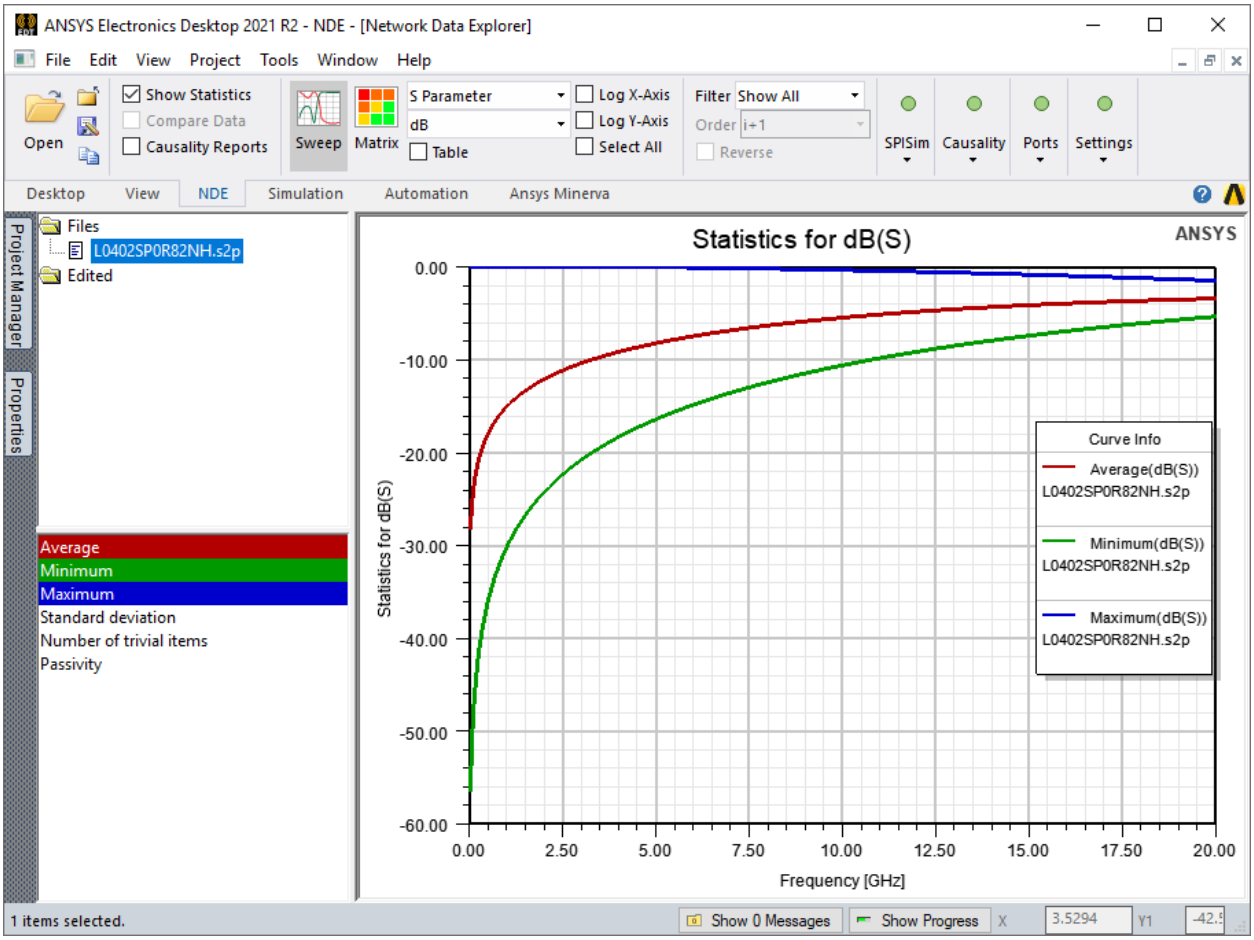
For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

Creating a Statistics Plot

Network Data Explorer can display a graph of selected statistical measures across all frequencies.

1. On the **NDE** ribbon:
    a. Click **Sweep**.
    b. Click **Show Statistics**.
2. In the Cell and Frequency Selection pane, select statistics to display. The selected statistics are plotted.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Hover the cursor over a statistic to view more information about it.

Passivity is only available for S-parameters (comparisons inactive).

For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

Comparing Network Data

Network Data Explorer can compare variations for two network data sets that are the same size.

1. On the **NDE** ribbon, click **Sweep**.
2. In the Network Data Selection pane, select exactly two data sets.
3. On the **NDE** ribbon, click **Compare Data**.
4. In the Cell and Frequency Selection pane, select which cells you want to compare . For each value along the X-axis, the Y-axis values are subtracted, one from the other, to create the comparison plot. The second selected data is subtracted from the first selected

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

data.

5. Optionally, check **Show Statistics** to show values applied to all cells and all frequencies

It is not possible to compare a data set against itself *unless the data set has been cloned.* Then, you can compare the original data set to the clone.

Traces for a given cell or statistical measure are displayed for all data sets; you can use tool tips to distinguish between them.

If a single cell or statistical measure is displayed, different colors are used for each data trace. If multiple cells or statistical measures are selected, a single color is used for all data traces for each cell or statistical measure.

In a data comparison, traces are shown for all selected data sets. This is true for both cell and statistical traces.

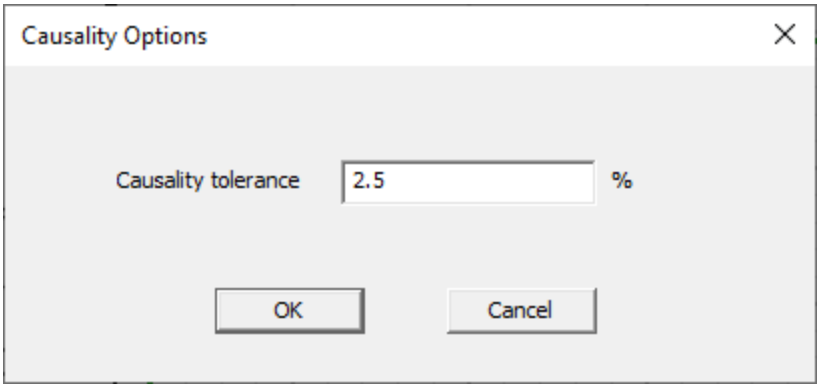
If **dB** format is shown, you can either subtract values before or after applying the dB function. See Display Format to make this choice.

Causality Checking and Plots

Network Data Explorer can perform a causality check on S-parameter data from any source (solution or file), and provide plots of the results in various formats.

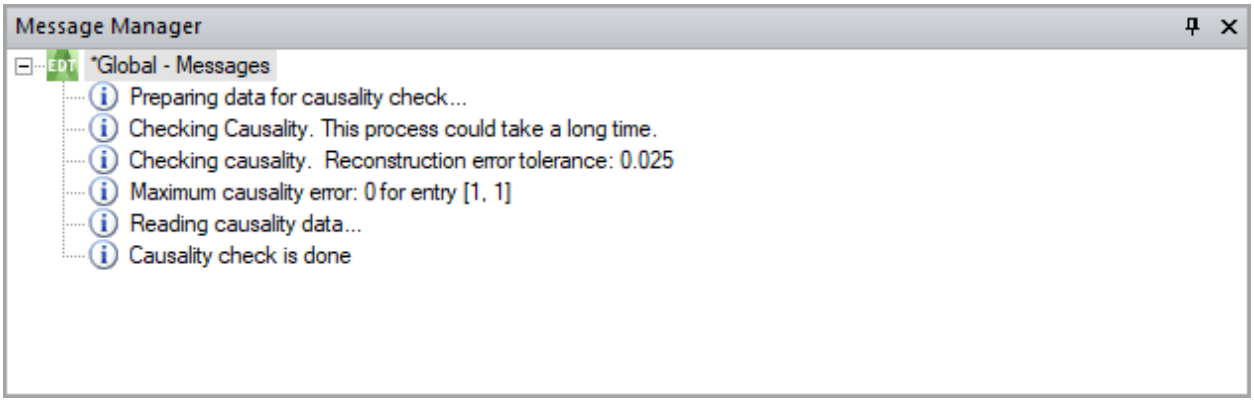
When S-parameter data is loaded into Network Data Explorer, the **Causality** button is enabled.

1. On the **NDE** ribbon, click **Causality**. The **Causality Options** dialog opens.



2. Enter a **Causality tolerance** and click **OK** to start the causality check. Depending on the size of the S-parameter data, the causality check may take several minutes to complete. The check's status displays in the Message Manager.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

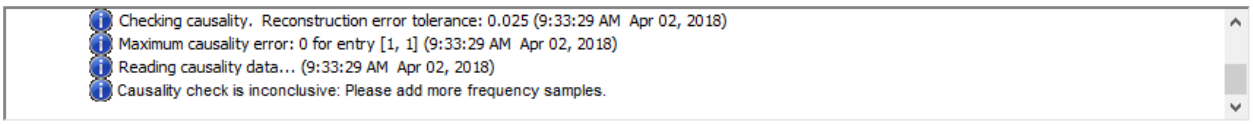


When the check completes, the Message Manager updates to display a summary of results.

Reconstruction Error Tolerance – Causality of a frequency response is determined by calculating the generalized Hilbert transform of the data at all frequencies. A causal frequency response is equal to its generalized Hilbert transform. The reconstruction error is the difference between the tabulated data and its transform at a given frequency. The message shows the maximum reconstruction error tolerance for a causal frequency response. The default tolerance of 0.01 is equal to the state-space fitting tolerance.

Maximum Causality Error – The maximum causality error for all port pairs and all frequencies, along with the matrix indices (port numbers) where the maximum noncausality occurs. A noncausal response is one where all matrix entries can be conclusively analyzed, and at least one entry exceeds the causality tolerance. The maximum reconstruction appears first, followed by port numbers in brackets (e.g., [port number, port number]). When all results are conclusive but no matrix reconstruction error exceeds the tolerance, the maximum causality error is reported as zero, and no matrix entry is listed.

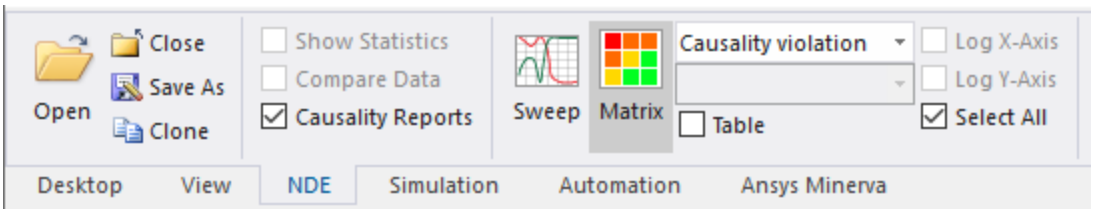
If the data does not contain enough frequency points to determine whether the data is or is not causal, the Messages Pane will note an inconclusive result. Network Data Explorer will also report the data set as inconclusive if any cells are inconclusive, even if other entries exhibit causality violations.



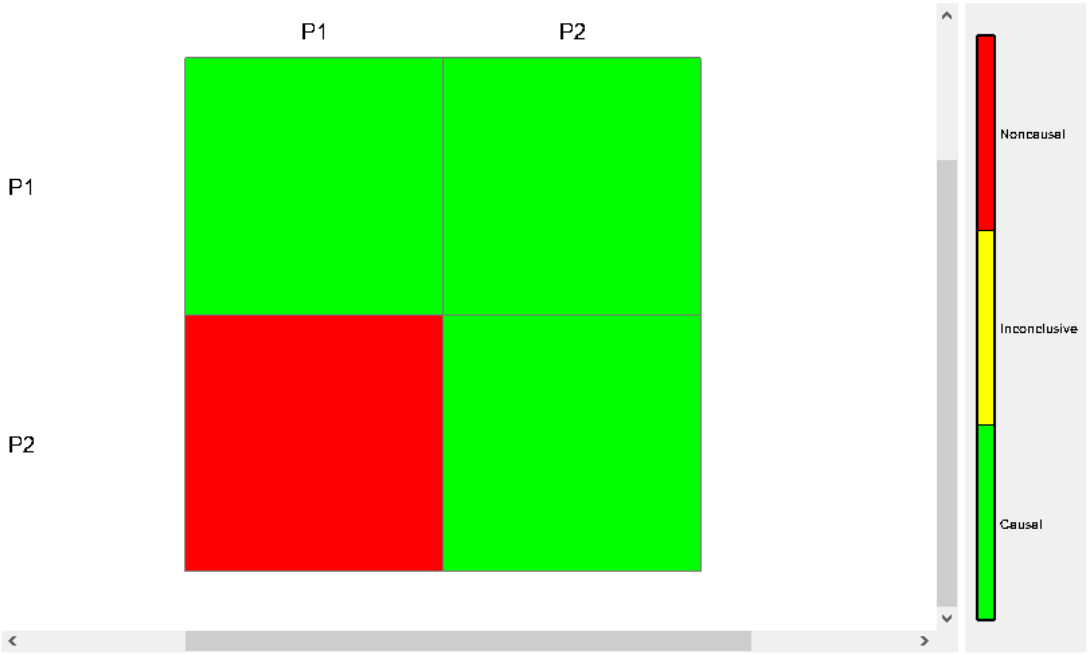
To see the plotted results of the causality check:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Matrix** icon or select **Causality violation** in the **Parameter** type 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.



A rectangular plot displays, with dimension N x N and color-coding to indicate the causality status of each port pairing.



In this plot, the cells go from (Port 1, Port 1) at the upper-left area to (Port N, Port N) at the lower-right corner. The result shows the causality over all frequencies in the data. In this example, the matrix is asymmetric, so that S12 is noncausal, while S11, S21, and S22 are causal.

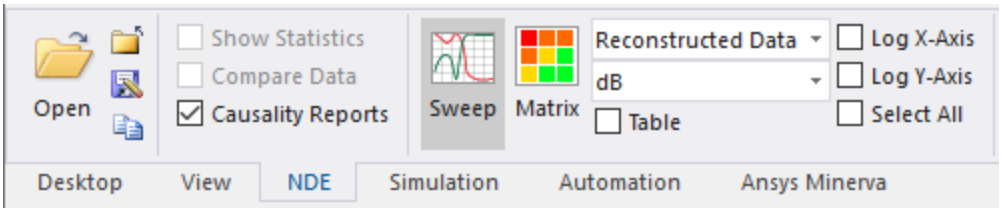
To see the details for each frequency, click the **Table** check button on the **NDE** ribbon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Freq		Port1	Port2	Port1	Port2
0.10000GHz	Port1	0.01715, 0.00000	0.00050, 0.00000	0.01493, 0.00000	0.00098, 0.00000
	Port2	0.53111, 0.00000	0.00579, 0.00000	0.11471, 0.00000	0.00832, 0.00000
0.20000GHz	Port1	0.01367, 0.00000	0.00059, 0.00000	0.00736, 0.00000	0.00052, 0.00000
	Port2	0.27237, 0.00000	0.00394, 0.00000	0.05933, 0.00000	0.00423, 0.00000
0.30000GHz	Port1	0.01362, 0.00000	0.00105, 0.00000	0.00344, 0.00000	0.00028, 0.00000
	Port2	0.24509, 0.00000	0.00239, 0.00000	0.01921, 0.00000	0.00195, 0.00000
0.40000GHz	Port1	0.00907, 0.00000	0.00103, 0.00000	0.00171, 0.00000	0.00006, 0.00000
	Port2	0.16665, 0.00000	0.00163, 0.00000	0.01530, 0.00000	0.00084, 0.00000
0.50000GHz	Port1	0.00693, 0.00000	0.00062, 0.00000	0.00075, 0.00000	0.00006, 0.00000
	Port2	0.11877, 0.00000	0.00106, 0.00000	0.00542, 0.00000	0.00040, 0.00000
0.60000GHz	Port1	0.00295, 0.00000	0.00044, 0.00000	0.00047, 0.00000	0.00005, 0.00000
	Port2	0.05535, 0.00000	0.00042, 0.00000	0.00462, 0.00000	0.00015, 0.00000
0.70000GHz	Port1	0.00015, 0.00000	0.00009, 0.00000	0.00029, 0.00000	0.00003, 0.00000
	Port2	0.00510, 0.00000	0.00012, 0.00000	0.00114, 0.00000	0.00008, 0.00000
0.80000GHz	Port1	0.00058, 0.00000	0.00068, 0.00000	0.00047, 0.00000	0.00004, 0.00000
	Port2	0.02772, 0.00000	0.00088, 0.00000	0.00115, 0.00000	0.00010, 0.00000
0.90000GHz	Port1	0.00210, 0.00000	0.00081, 0.00000	0.00040, 0.00000	0.00013, 0.00000
	Port2	0.05492, 0.00000	0.00133, 0.00000	0.00173, 0.00000	0.00010, 0.00000
1.00000GHz	Port1	0.00221, 0.00000	0.00064, 0.00000	0.00033, 0.00000	0.00010, 0.00000
	Port2	0.07703, 0.00000	0.00142, 0.00000	0.00111, 0.00000	0.00009, 0.00000
1.10000GHz	Port1	0.00410, 0.00000	0.00045, 0.00000	0.00037, 0.00000	0.00008, 0.00000
	Port2	0.08851, 0.00000	0.00151, 0.00000	0.00137, 0.00000	0.00011, 0.00000
1.20000GHz	Port1	0.00433, 0.00000	0.00078, 0.00000	0.00017, 0.00000	0.00013, 0.00000
	Port2	0.09013, 0.00000	0.00100, 0.00000	0.00168, 0.00000	0.00017, 0.00000
1.30000GHz	Port1	0.00377, 0.00000	0.00072, 0.00000	0.00020, 0.00000	0.00018, 0.00000
	Port2	0.09487, 0.00000	0.00151, 0.00000	0.00081, 0.00000	0.00012, 0.00000
1.40000GHz	Port1	0.00380, 0.00000	0.00035, 0.00000	0.00020, 0.00000	0.00014, 0.00000
	Port2	0.08918, 0.00000	0.00136, 0.00000	0.00139, 0.00000	0.00009, 0.00000
1.50000GHz	Port1	0.00417, 0.00000	0.00005, 0.00000	0.00012, 0.00000	0.00011, 0.00000
	Port2	0.08091, 0.00000	0.00174, 0.00000	0.00094, 0.00000	0.00021, 0.00000
1.60000GHz	Port1	0.00315, 0.00000	0.00038, 0.00000	0.00006, 0.00000	0.00015, 0.00000
	Port2	0.07512, 0.00000	0.00208, 0.00000	0.00065, 0.00000	0.00011, 0.00000
1.70000GHz	Port1	0.00275, 0.00000	0.00034, 0.00000	0.00006, 0.00000	0.00021, 0.00000
	Port2	0.06069, 0.00000	0.00152, 0.00000	0.00101, 0.00000	0.00021, 0.00000

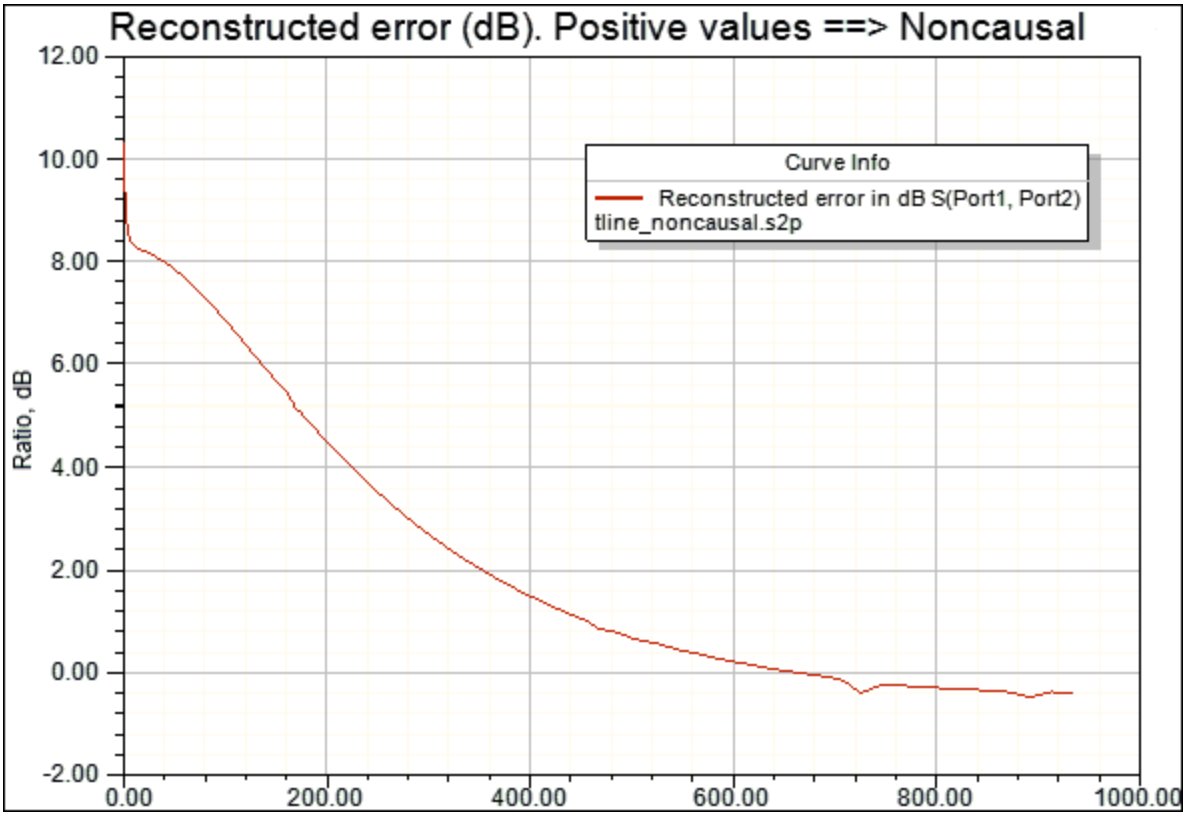
To plot the reconstructed frequency response generated by the causality checker:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Sweep** icon or select **Reconstructed Data** in the **Parameter** type drop-down menu.
3. In the **Format** drop-down menu, select the desired format. **dB** is selected by default.



The plot appears, showing the reconstruction error at each frequency divided by the tolerance.

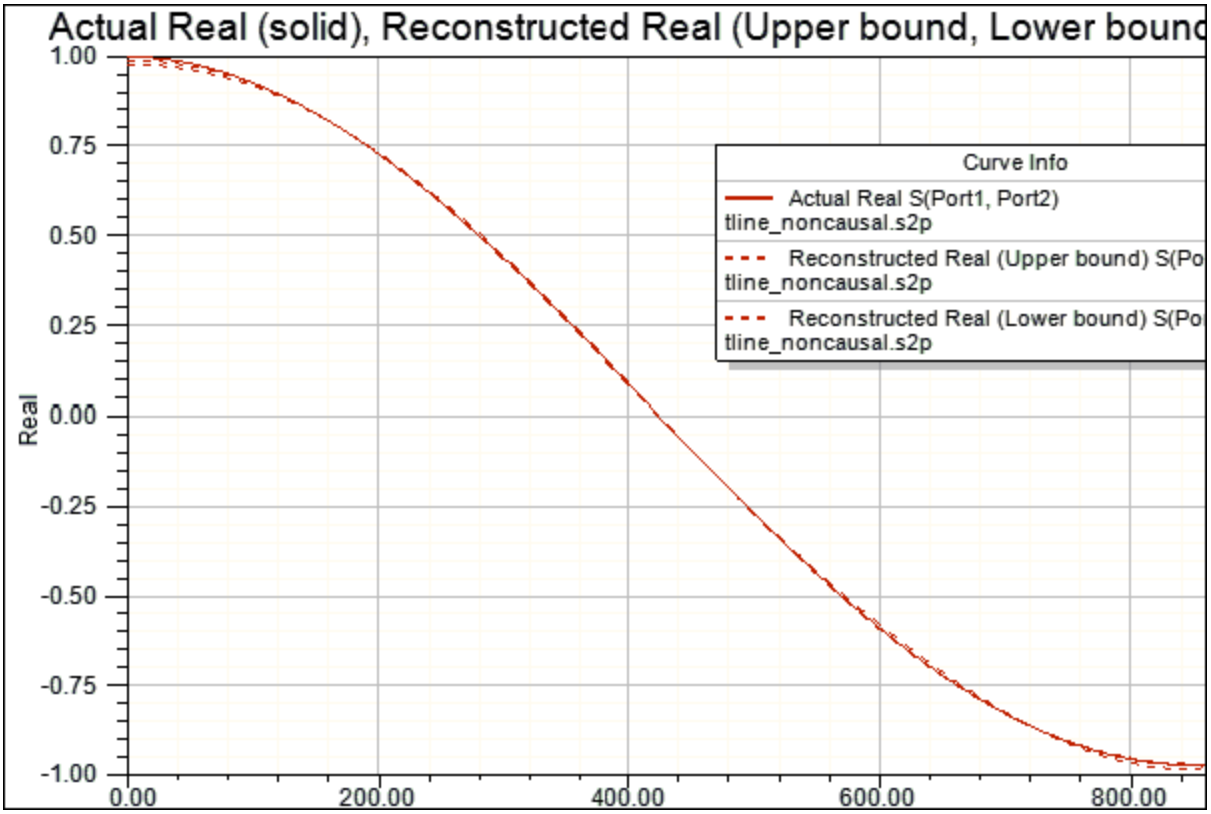
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The reconstruction error ratio for parameter S12 is positive for frequencies less than about 680 MHz, indicating a broad range of noncausal behavior.

4. To compare the real part of the reconstructed data to the real part of the actual data, set the **Format** to **Real**.

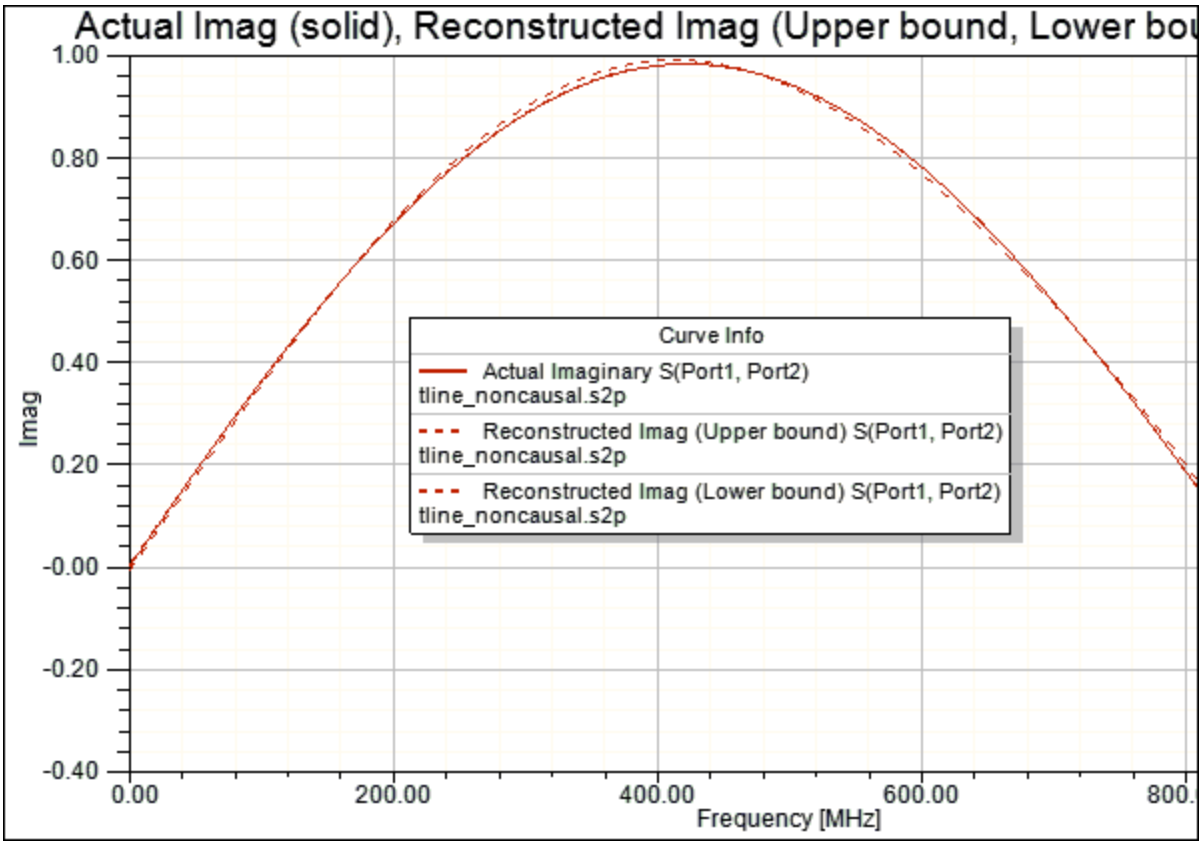
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

5. To compare the imaginary part of the reconstructed data to the imaginary part of the actual data, set the **Format** to **Imaginary**.

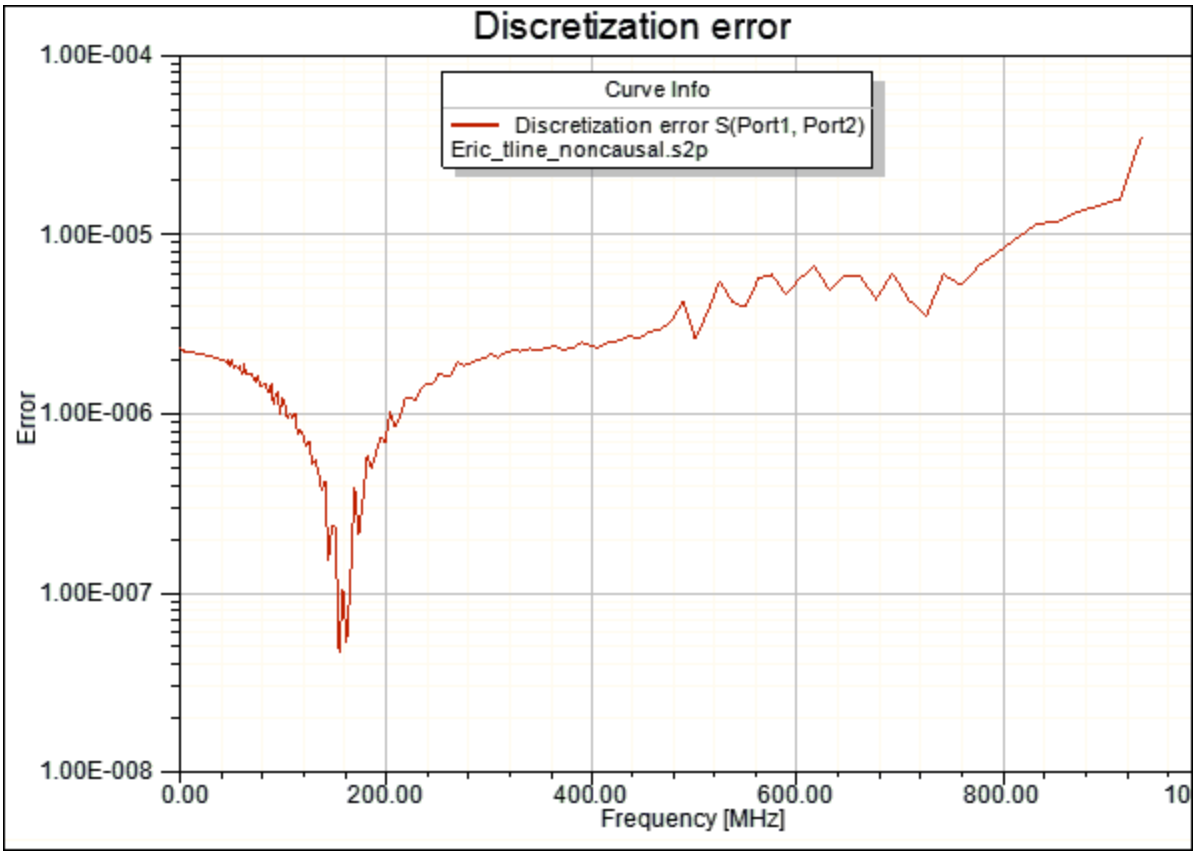
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

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 view the frequency-dependent discretization error, set the **Format** to **Discretization**.



The discretization error is the error that is due to the fact that the data are available only at discrete frequencies rather than for a continuous spectrum. A discretization error near or greater than the causality tolerance renders the causality check inconclusive. Data at more frequencies could reduce the discretization error and render the analysis conclusive. This set of data exhibits low discretization errors (<<0.01) at all frequencies, and the causality check is conclusive (conclusively noncausal in this example).

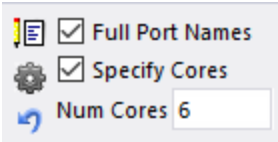
Multithreading

By default, multithreading (execution on multiple cores) is enabled for Causality Checking and Macro Model export. Multithreading saves significant time in the Causality Check calculation, and improves the time for other state-space fitting operations. See Technical Notes.

The **Cores** field in the Network Data Explorer Control Pane defaults to half the number of cores detected on your computer.

For best performance, disable hyper-threading on your computer. When hyper-threading is enabled, the number of cores includes the physical cores and an equal number of logical cores. With hyper-threading disabled, the display shows only half the number of physical cores.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To disable multithreading, uncheck the check box.

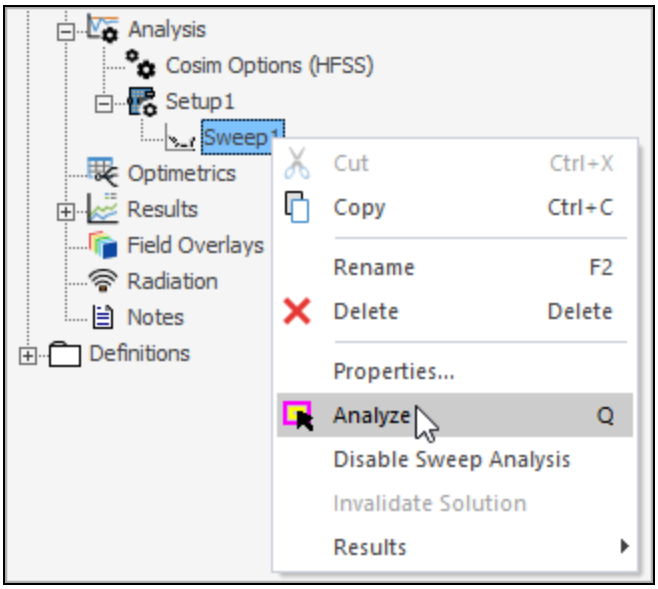
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15 - Running Simulations

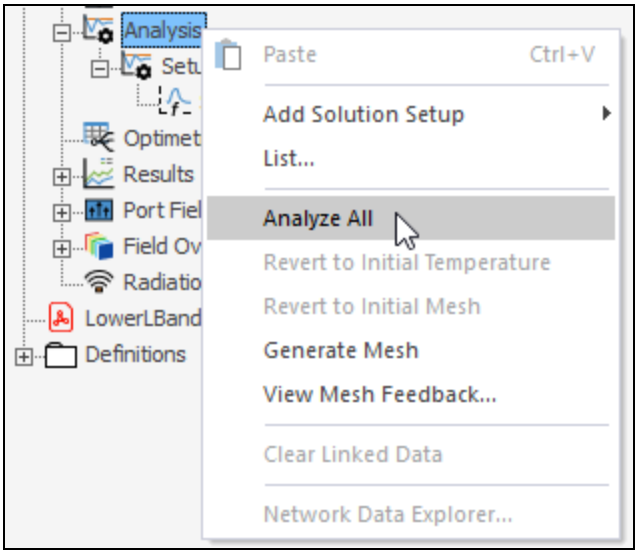
After you specify how Ansys Electronics Desktop is to compute the solution, you need to begin the solution process. You can access the **Analyze** commands for a specific setup by right-clicking on a solution or sweep in the **Project** tree and selecting from the shortcut menu, you by selecting the **Analyze All** from the solver menu, or the **Analyze All** icon on the **Simulation** tab of the ribbon.

In general, the **Analyze** command on the shortcut menu applies to the selected setup and associated sweeps, if any, or to a selected sweep. To use this command, right-click on a setup or sweep in the Project Manager and click the command on the context menu.



The **Analyze All** command applies to all enabled setups, dependent setups, and sweeps at or below the level invoked in the Project Manager. To use this command, either click **HFSS 3D Layout** > **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.
- one or more solution setups or sweeps.
- Submit Job to RSM or HPC Scheduler
- Monitor queued simulations
- Change a solution priority for system resources
- Abort an analysis
- Re-solve after modifying a design

Solving a Single Setup

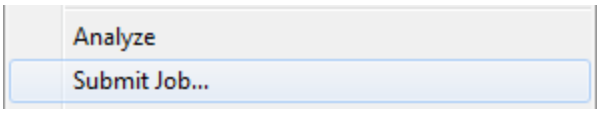
To run a single setup:

1. Select a solution setup in the Project Manager.
2. Right-click and select **Analyze** from the shortcut menu. The graphic shows a single selected for analysis.

   The 3D field solution is computed inside the structure for a solution.

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 right-click a Setup the shortcut menu also includes the **Submit Job...** command.



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 **HFSS 3D Layout** > **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**.

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.

To view the *Solutions* window:

1. Right-click the solution **Setup** in the Project Manager.

   A shortcut menu appears.

2. Select **Convergence**, **Solutions**, **MatrixData**, **Mesh Statistics**, or **Profile** from the shortcut menu.

   The *Solutions* window appears with the corresponding tab selected and the current data displayed.

To view the solution status:

-

  The *Solutions* dialog box appears with the **Browse** tab selected. It contains a tree structure showing the solutions listed according to Setup, Solution, and Variation. A table lists the setup, the solution, and the state of the solution.

- You can use the **Properties** button to display a dialog box that lets you change the way the Setup, Solution, and Variation are listed in the tree structure of the *Solutions* dialog box.
- You can delete one or more solutions by selecting from the table and clicking **Delete**. Click on a solution to select it, and use Ctrl+click to select multiple solutions, or Shift+click to select a range of solutions. You can also select all solutions using the **Select All** button.

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 license is lost, the software waits for the license to be regained, checking every 2 minutes or until you abort.

- The **Statistics** tab of the *Solutions* dialog box displays path information as well as format, number of files, and size.

Changing a Solution Priority for System Resources

You can modify the priority of Ansys Electronics Desktop simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of Ansys Electronics Desktop simulations, your other software tools will respond as they normally would, but Ansys Electronics Desktop simulations may take longer.

> **Note:**
>
> The Windows Task Manager does not indicate a reduced priority for the Ansys Electronics Desktop solvers. It only lists the priority of the engine manager, which appears normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations for the system's resources:

1. While a solution is running, right-click the **Progress** window, and click **Change Priority** on the shortcut menu.
   - To affect priority for future simulation runs, click **Tools** > **Options** > **HPC and Analysis** and click the **Options** tab of the resulting dialog box.
2. From the **Change Priority** menu (or the **Default Process Priority** drop-down menu), select one of the following priorities:

   Highest
   Above Normal
   Normal Default
   Below Normal
   Lowest Priority

3. Click **OK**.

Related Topics

Monitoring Queued Simulations

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 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 has been solved.

Re-solving after Modifying a Design

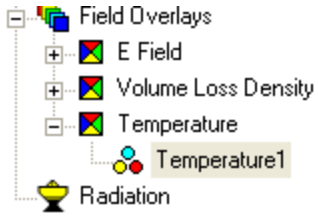
In some cases, if you modify a design after generating a solution, the solution in memory will no longer match the design. In such cases you receive a warning message that "Solutions have been invalidated. Undo to recover."

To generate a new solution after modifying a design, follow the procedure for running a simulation.

Re-solving after Ansys Workbench Thermal Feedback

With the Enable Feedback box in Setting the Temperature of Objects dialog box is checked, you can manage analysis with feedback in Ansys Workbench. After solving an HFSS or Q3D Extractor or Maxwell design, after performing the corresponding linked thermal analysis in Ansys Workbench, you can receive a temperature distribution back from the thermal solution. Ansys Workbench will write the feedback files directly to the Project Solution directory.

After an analysis that includes thermal feedback from Ansys Workbench, you can see temperature changes expressed in Temperature field overlays (both visually in the overlay and in the color key) as well as in the Solution 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 **Solution** data Profile tab you will see a new entry for Maximum Delta T, for the change in temperature from the previous simulation. The solver calculates delta in the first iteration by comparing the temperature distribution output from thermal with the initial temperature setting in HFSS/Maxwell/Q3D. Subsequent simulation iterations provide a number for the temperature delta.

Solver MCS4	00:00:00	00:00:00	31.4 M	Disk = 0 KBytes, matrix size 3468 , matrix bandwidth
Field Recovery	00:00:00	00:00:00	31.4 M	Disk = 310 KBytes, 2 excitations
				Maximum Delta T = 7.0685

This simulation feedback loop from Ansys Electronics Desktop to Ansys Workbench and back can continue until you decide that Temperature delta reported in the Solution Report low and stable for the designs.

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 - 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***.

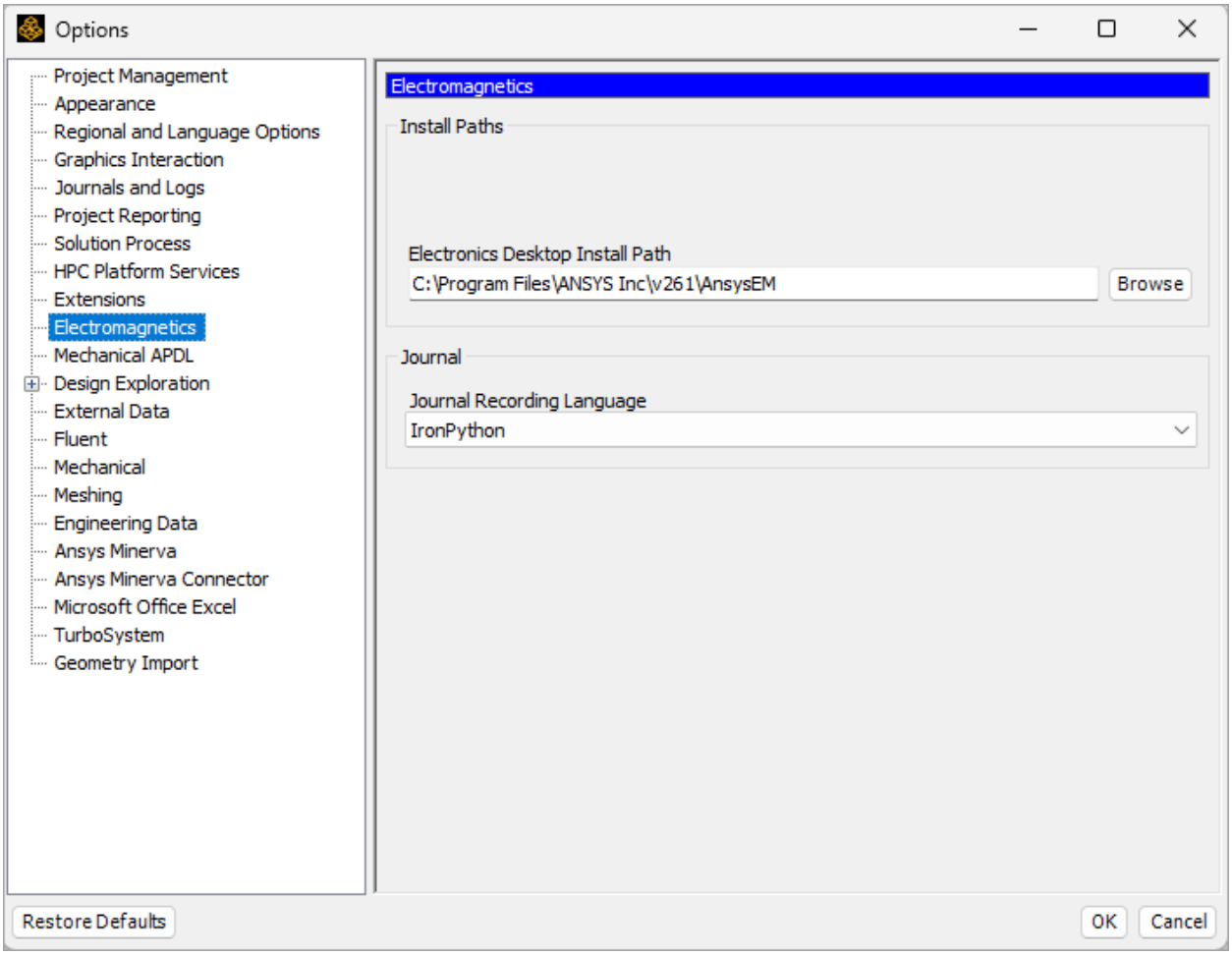
   A *Modify Integration with Ansys 20yy Rn* command window appears.

   - 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.

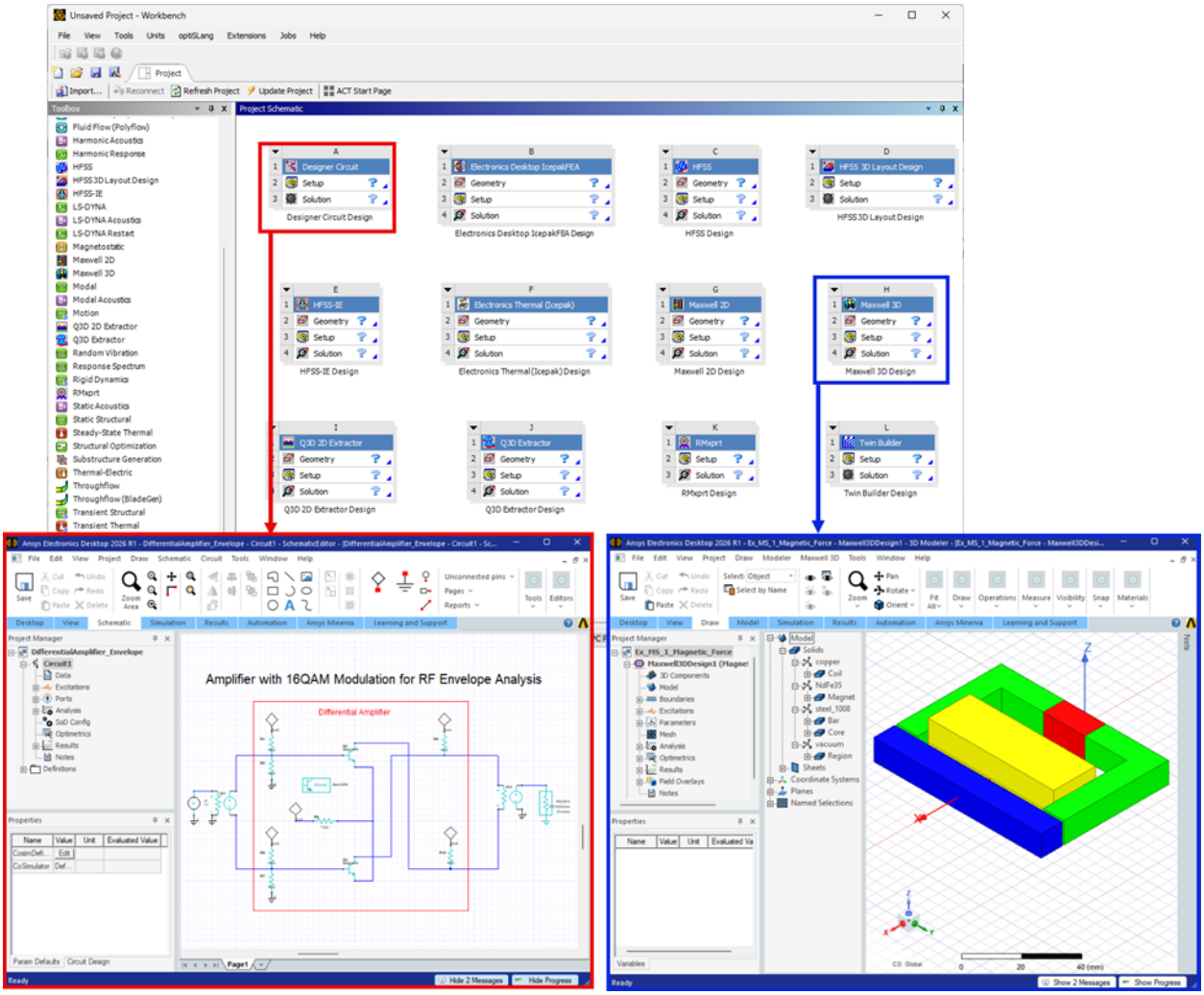


In this group of options, you can also specify Workbench's default **Journal Recording Language** for use when interoperating with the Ansys Electromagnetics products. **Iron Python** is the default language, and the alternative choice is **Visual Basic** (VB). Changes applied here affect any new Workbench 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.

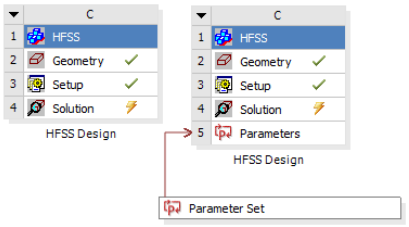


Objects placed on a Workbench Project Schematic (such as instances of Ansys Electromagnetics projects) are referred to as *systems*. The number of cells in the system depend on the design type, as follows:

- **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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

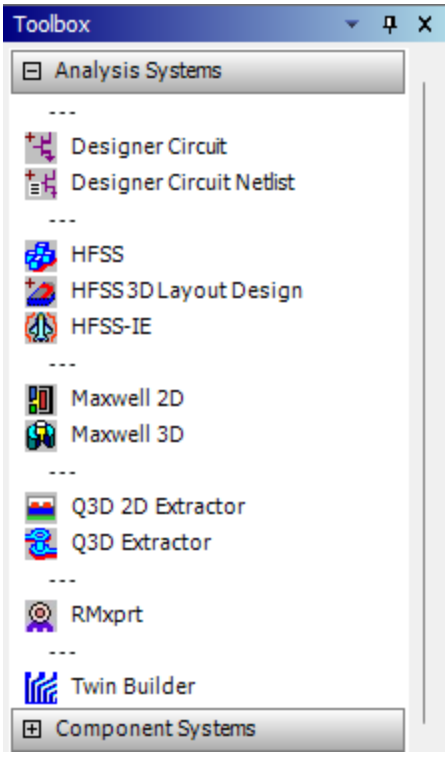
In addition to these major features, Workbench can archive, save, back up, duplicate, and delete Ansys Electromagnetics projects used in a Workbench project. Progress information and messages from integrated Ansys EM projects are also displayed in Workbench.

> **Note:**
>
> Look in the *Ansys Workbench Help* to find detailed information about performing the above-listed operations.

Adding New Analysis Systems

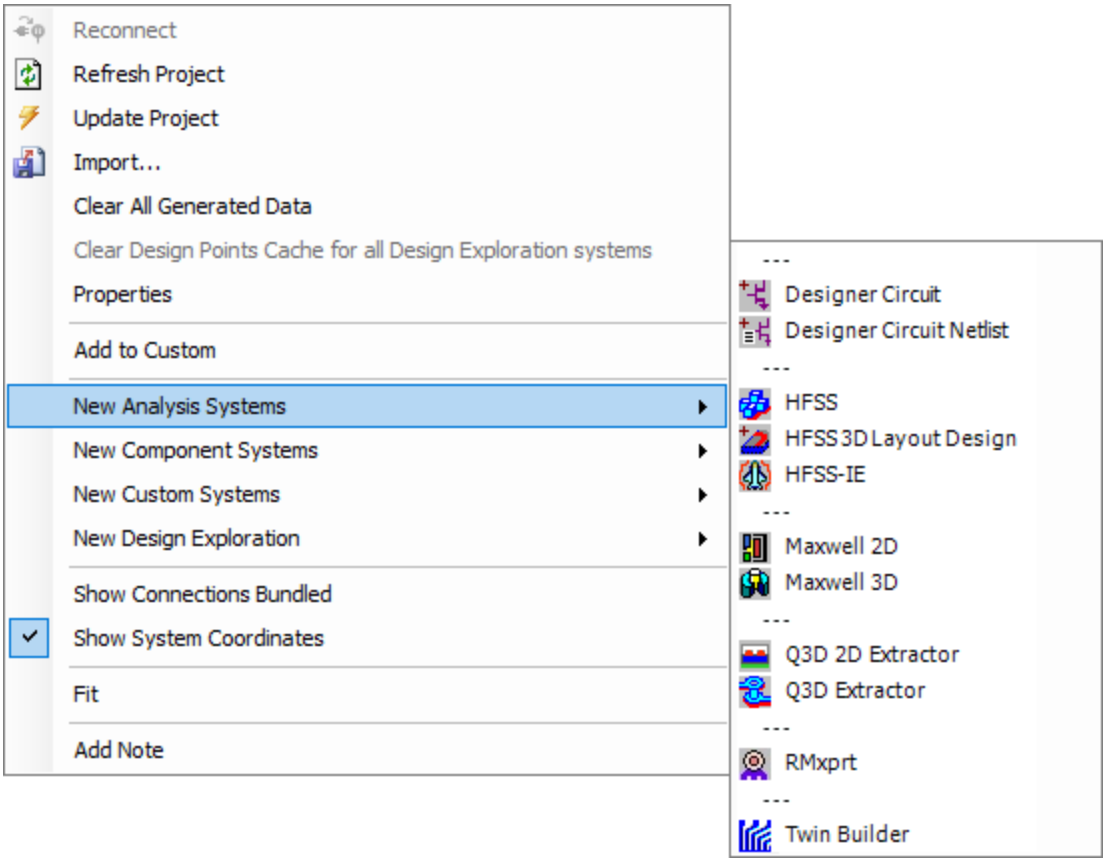
A new Ansys Electromagnetics Analysis System can be added to a Workbench Project Schematic by dragging and dropping it from the Toolbox:

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.

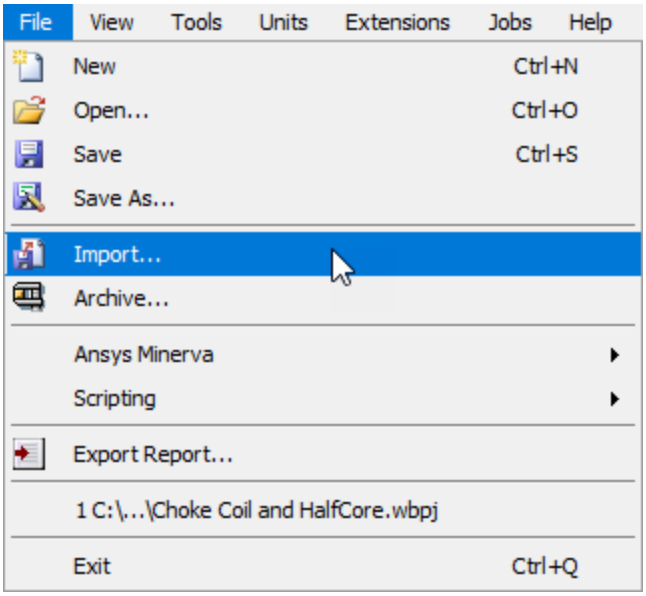


Importing Existing Ansys EM Projects into Workbench

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.

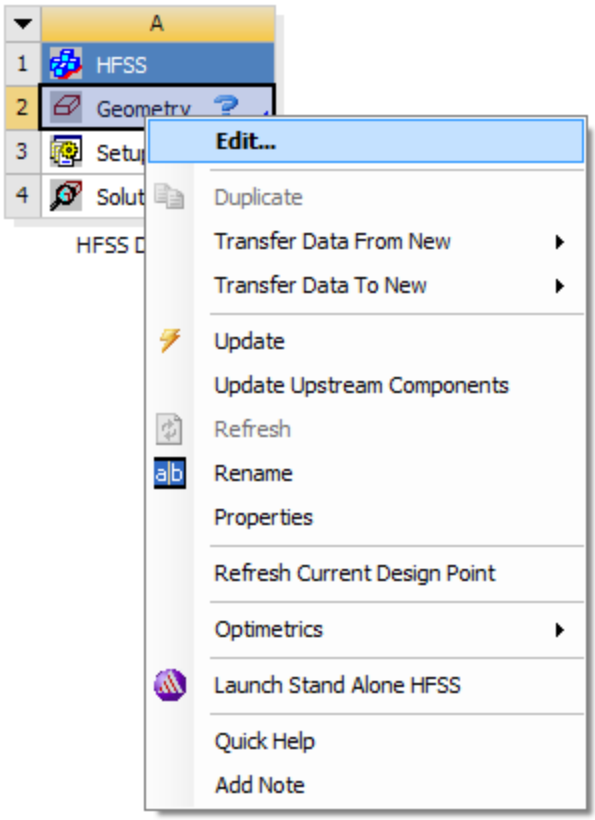


Editing Ansys EM Models in Workbench

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.

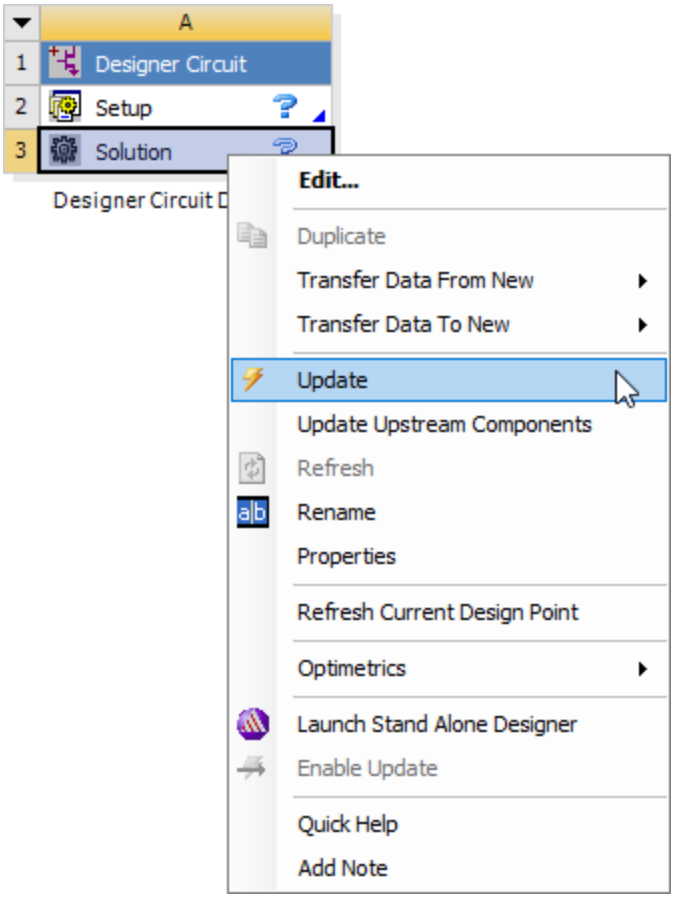
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



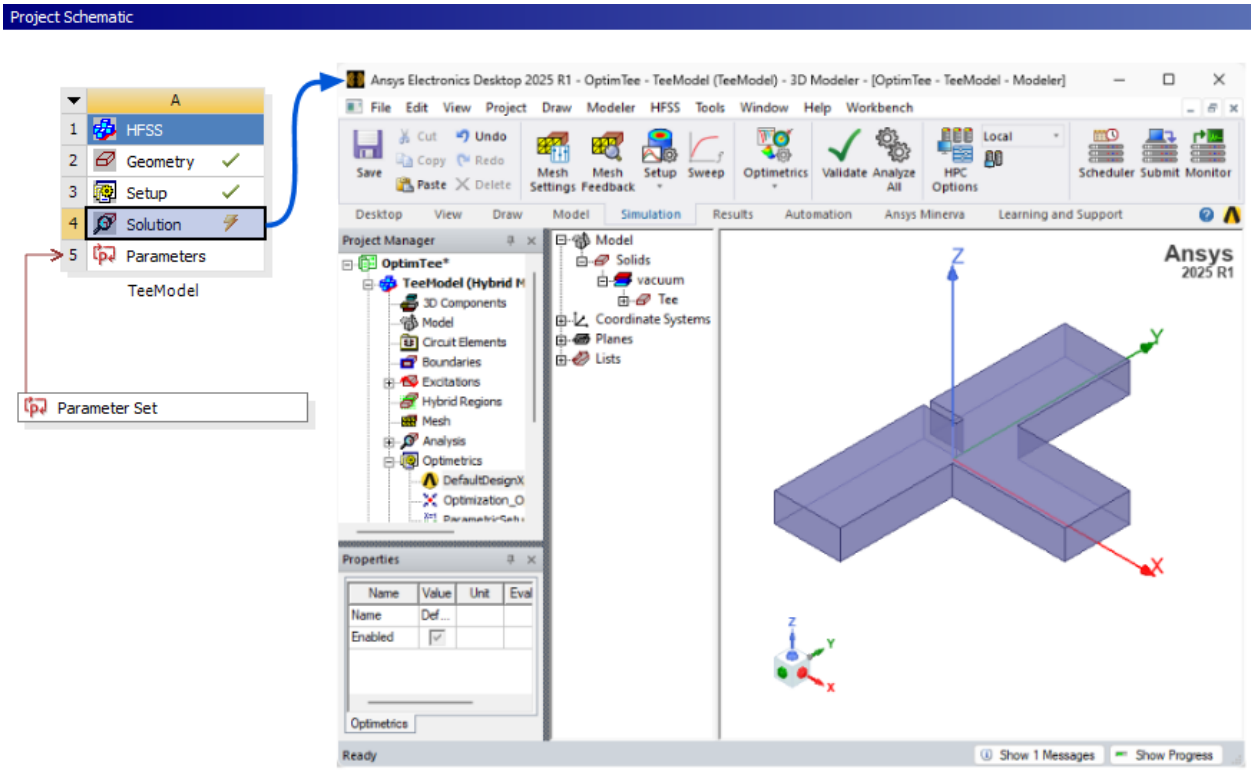
Analyzing Ansys EM Models in Workbench

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.



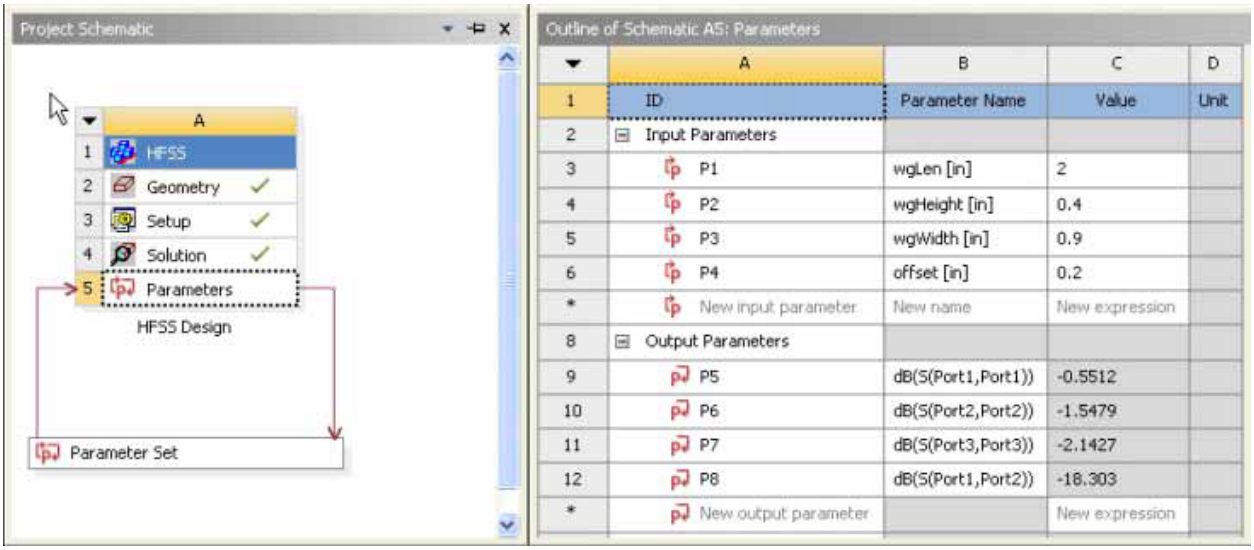
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

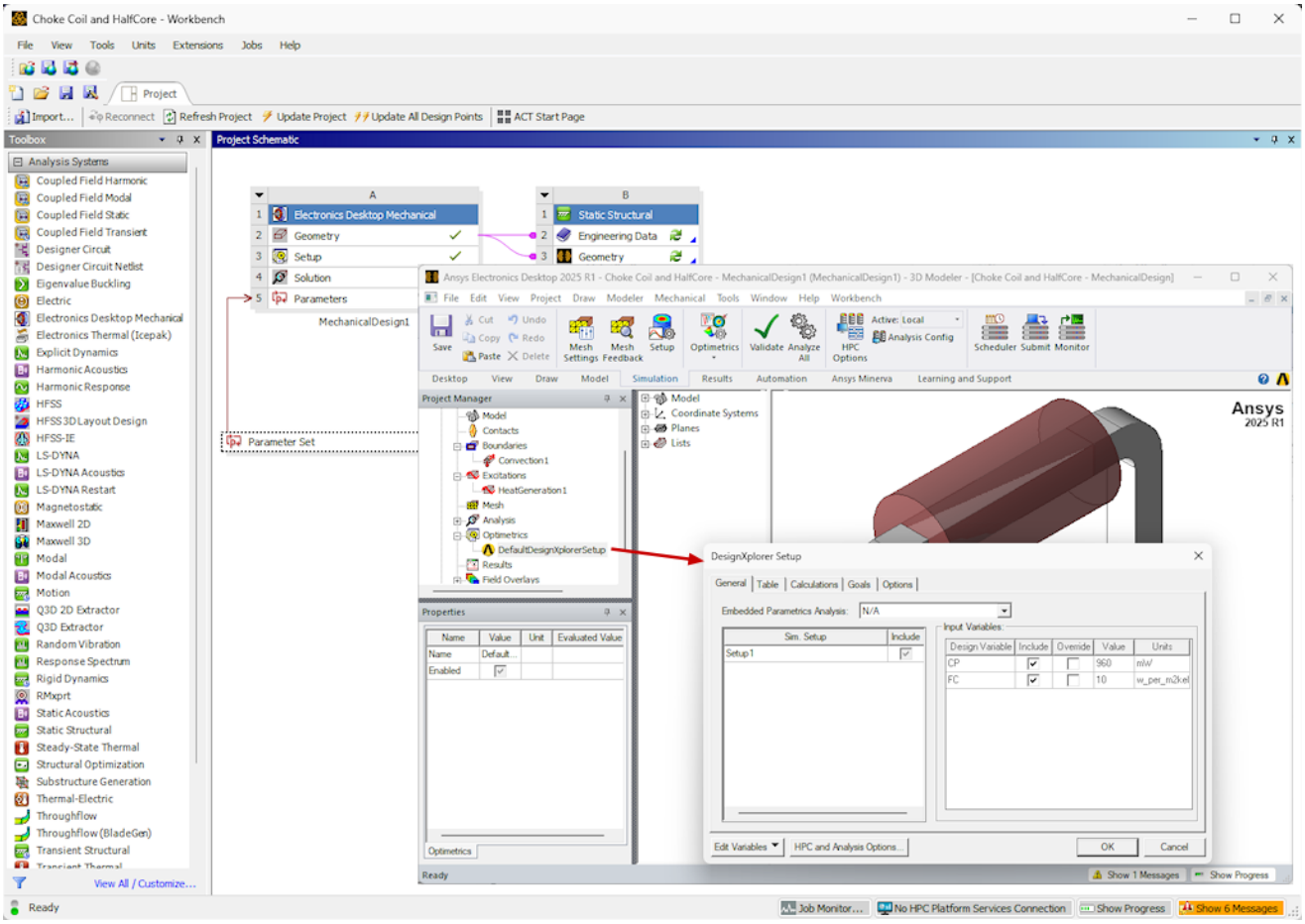


Note:

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:

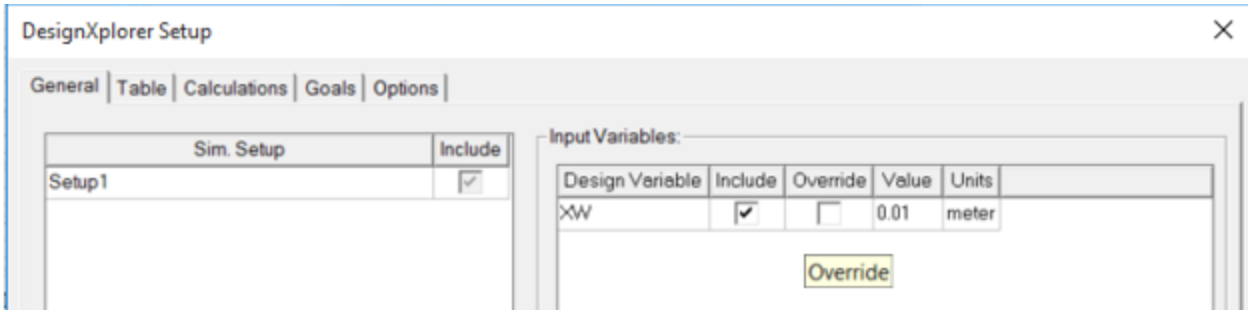
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



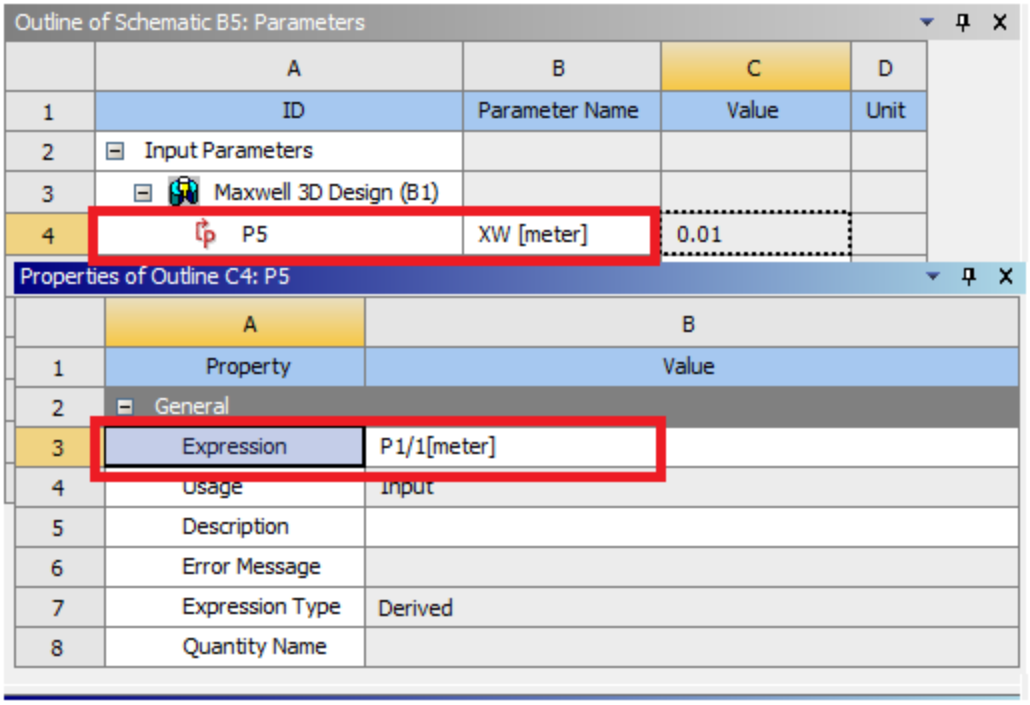
The workflow for using Electronics Desktop systems with the Workbench to take advantage of Distributed Analysis is as follows:

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.

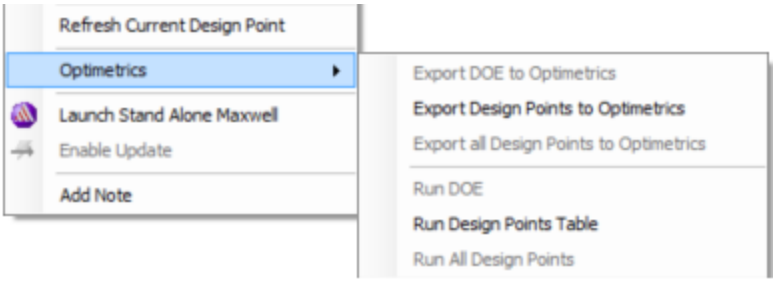


2. In the Workbench Design, change the P5 expression to a new value (in this example figure, the P5 parameter is assigned an expression of P1/1[meter] to tie it to the geometry system's variable.



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.

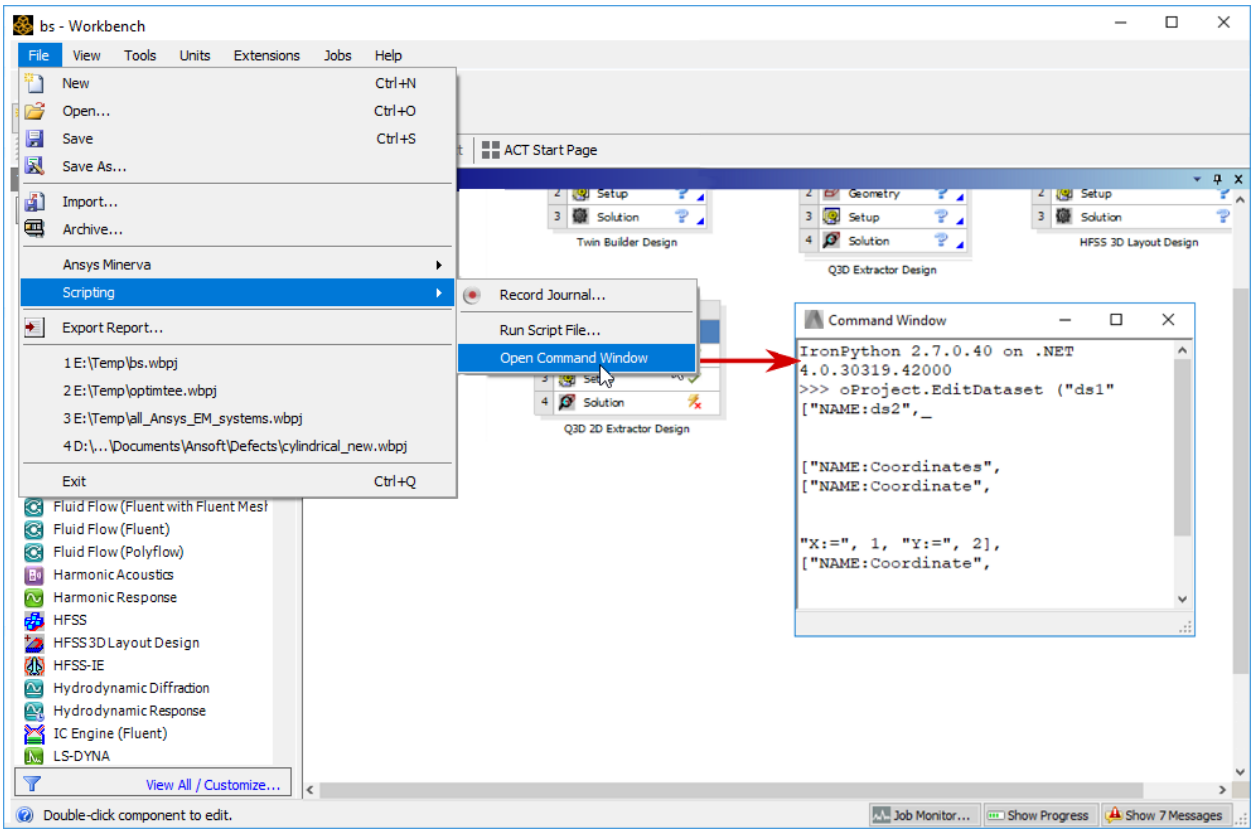


3. In Electronics Desktop, you can click on the default DesignXplorer Setup, and choose **Generate Variation Data**.

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 - Ansys Multiphysics Coupling

Data integration provides an improved multiphysics workflow between Ansys Electromagnetics designs and Ansys applications such as Mechanical and Thermal. Coupling is provided through project schematic links. Heat losses and force data are automatically transferred to Ansys Mechanical, and there is no need to export or import transfer XML files. Edits made in Ansys Electromagnetics applications are automatically transferred to the Ansys application through the Workbench **Refresh** command. Workbench commands also enable easier automation of iterative coupling of thermal feedback.

The following sections provide some examples of multiphysics coupling.

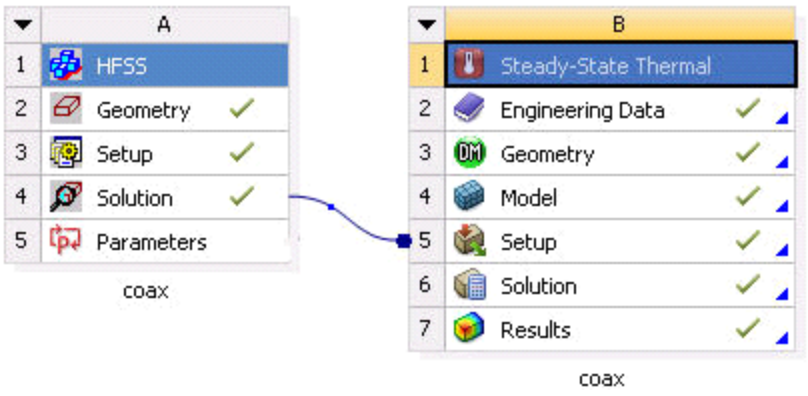
Multiphysics Coupling on Workbench with Ansys Thermal

Multiphysics Coupling on Workbench with Ansys Structural

Multiphysics Coupling on Workbench with Ansys Thermal

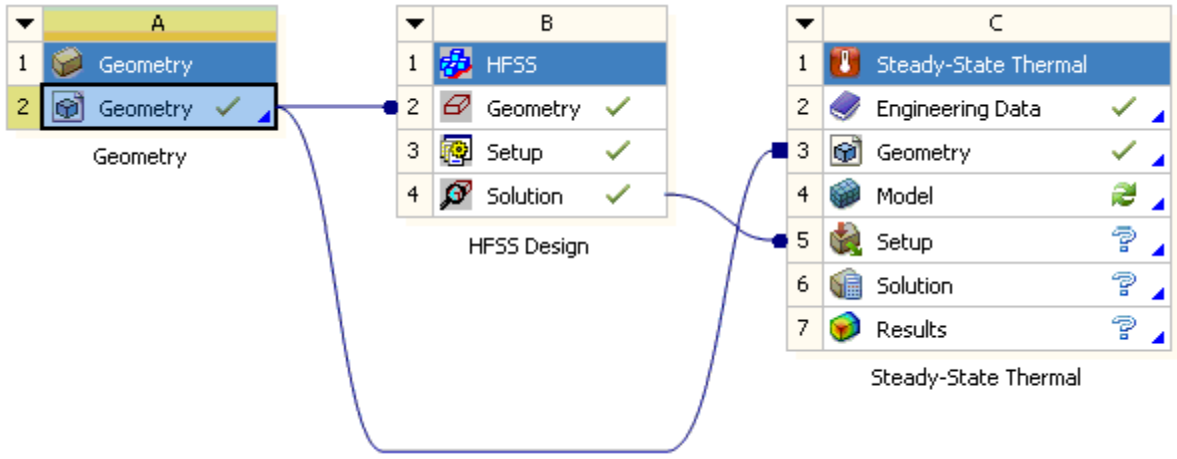
Using data integration, HFSS, Maxwell, and Q3D Extractor provide heat losses (heat generation and heat flux) to Ansys Thermal. In HFSS, both Driven and Eigenmode projects can be coupled for multiphysics. You first need to enable feedback as described in Setting the Temperature of Objects.

Note how the HFSS design is linked visually to Ansys Thermal on the Workbench project schematic.

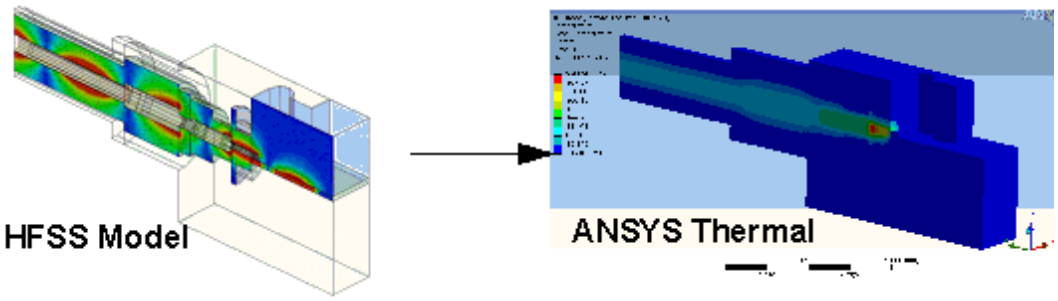


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry sharing is possible, provided you activate Beta Options in Workbench. A geometry from the Component Systems in the Toolbox in Workbench Schematic can be shared as shown in the image below.



In this example, HFSS coax model Solution provides heat loss data as a thermal load to the Ansys Thermal Setup. The resulting analysis shows a thermal "hotspot", providing the user with the information needed to adjust the design's material to fix the problem.

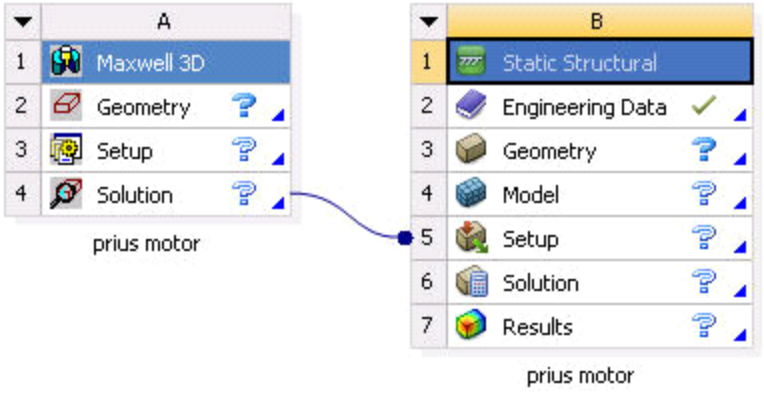


You enable this feature by checking Enable Feedback in the dialog for Setting the Temperature of Objects.

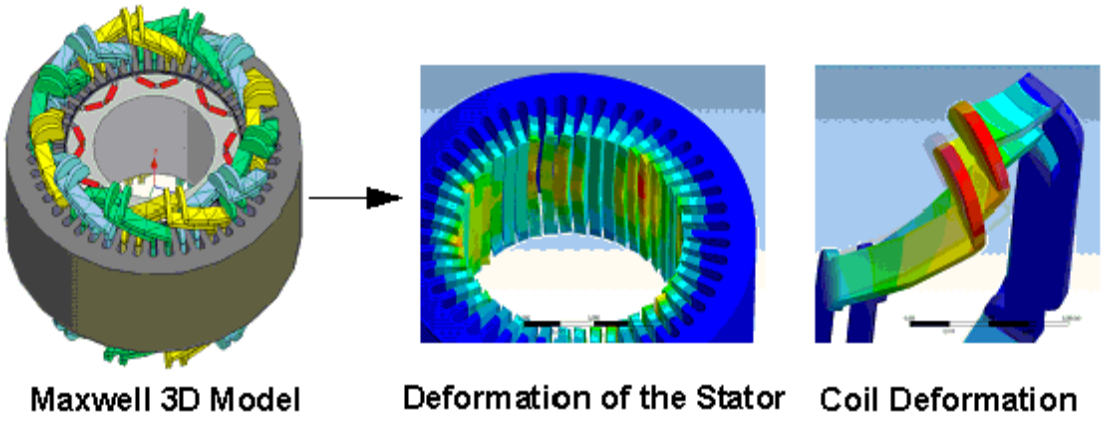
Multiphysics Coupling on Workbench with Ansys Structural

Using data integration, Maxwell 2D and Maxwell 3D can provide forces to Ansys Structural.

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 example, the Maxwell 3D electromagnetic force density **Solution** is used as the load in Ansys Structural to determine how these forces deform the motor's stator and coils.

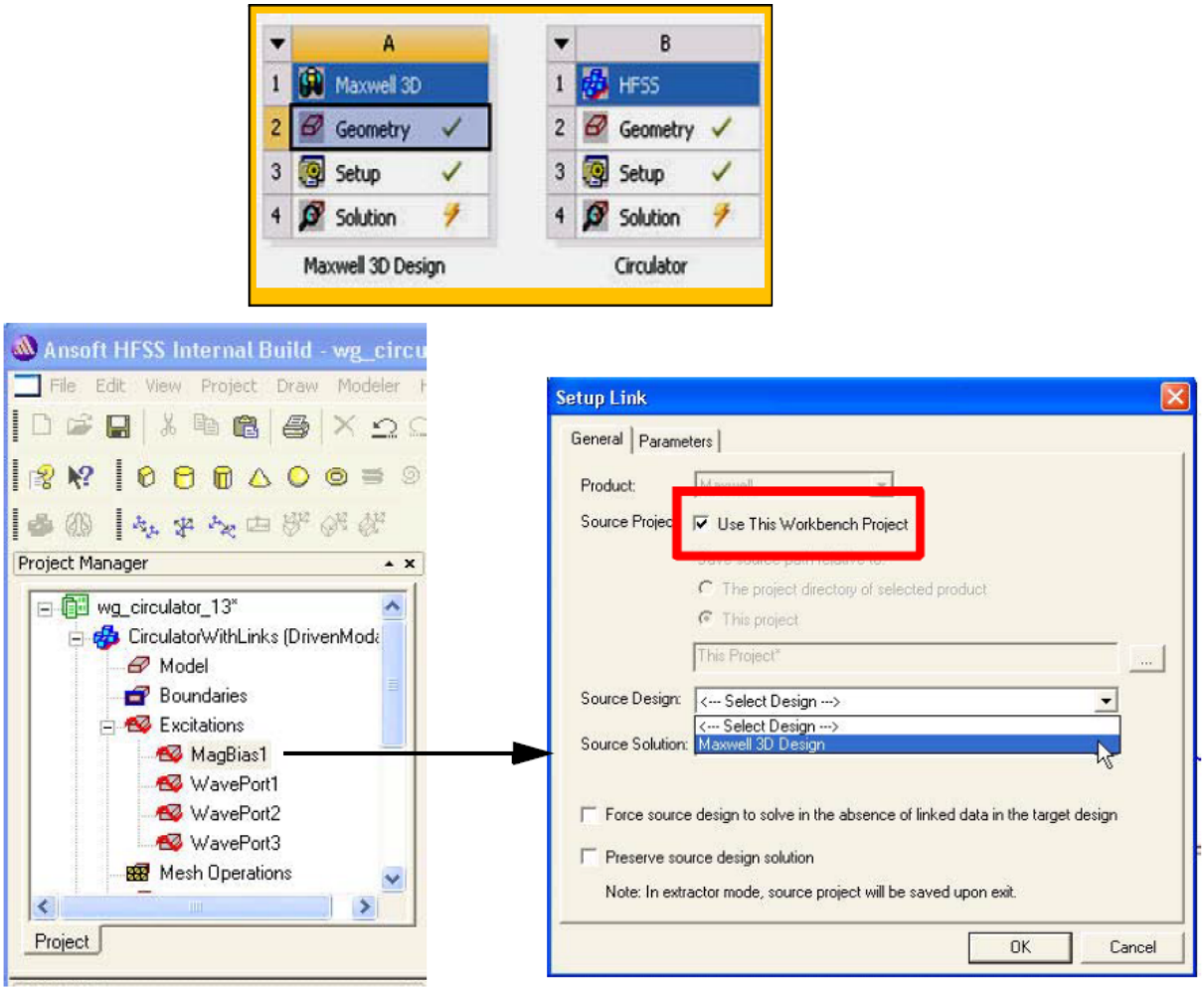


Multiphysics Coupling between Ansys EM Field Systems on Workbench

You can set up links between Ansys Electromagnetics field systems that reside on a Workbench project schematic. Linking is set up in the Ansys Electromagnetics application as shown in the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following example.

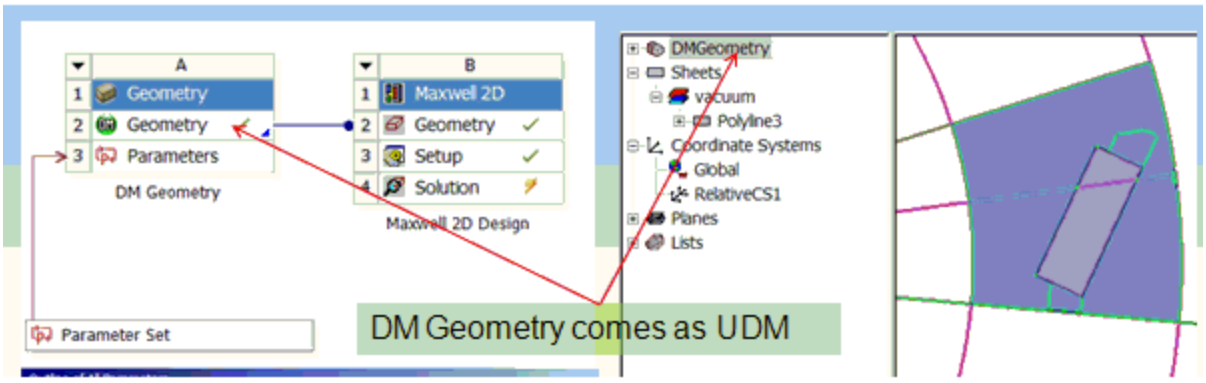


Ansys EM CAD Integration through Workbench

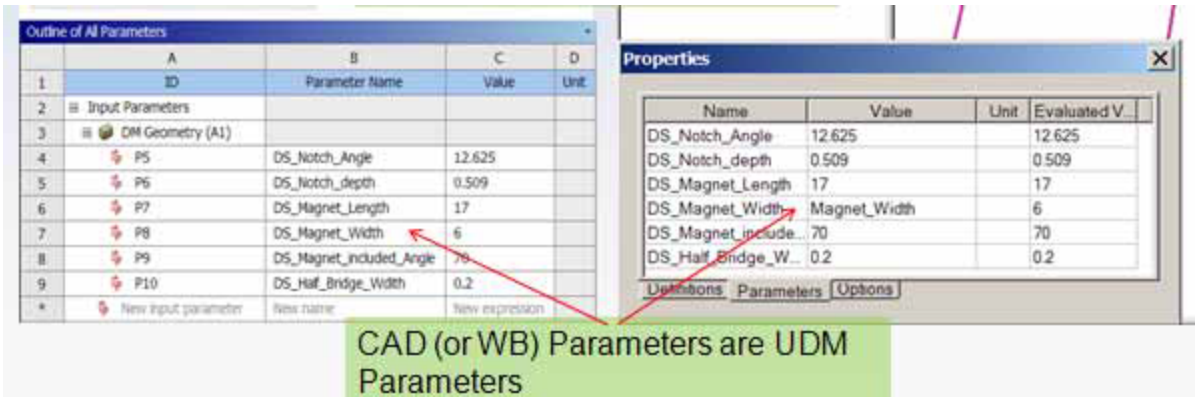
Ansys Electromagnetics CAD integration is a Workbench feature available for Ansys Electromagnetics 3D Products - HFSS, Maxwell, Q3D, Icepak, and IcepakFEA, as the Ansys Framework. The feature is available only through Workbench, and is not available via standalone Ansys Electromagnetics products.

Ansys CAD integration provides a bi-directional dynamic link through Workbench. This makes it possible to get updated geometry from CAD, modify CAD parameters in Ansys Electromagnetics products, and return an updated geometry. The feature is non-associative due to a need to reassign boundaries if a modified CAD model is used. The process creates a User Defined Model (UDM) for each geometry source.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



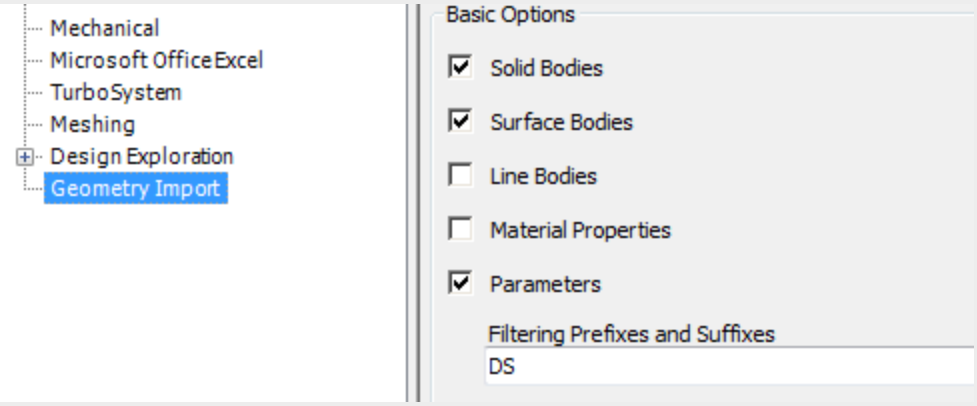
The UDM format makes it possible to exchange parameters. Ansys online help contains further description of the UDM feature and function.



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

Creating Dynamic Links to EDT Designs

Adding a Design Variable for Dynamic Links

CAD Integration Functionality

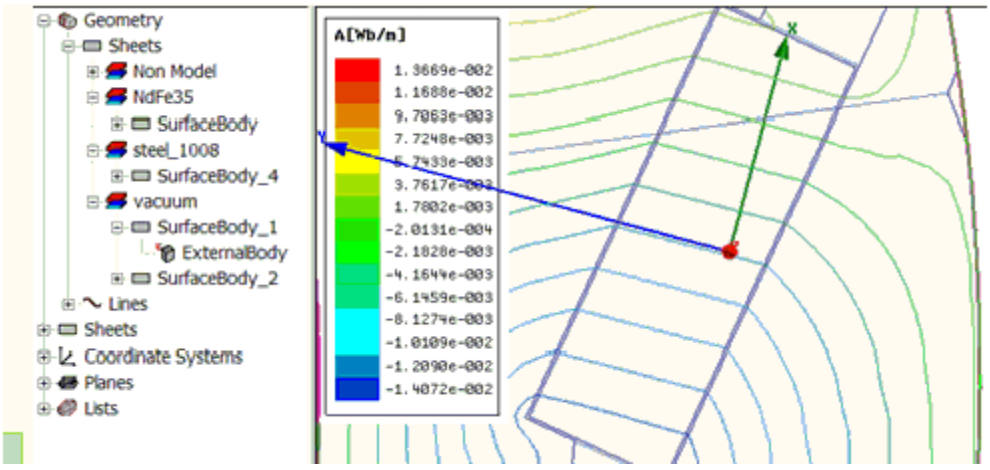
Healing with CAD Integration

Important Geometry Options for CAD Integration

CAD Integration Functionality

CAD Integration includes the following functionality:

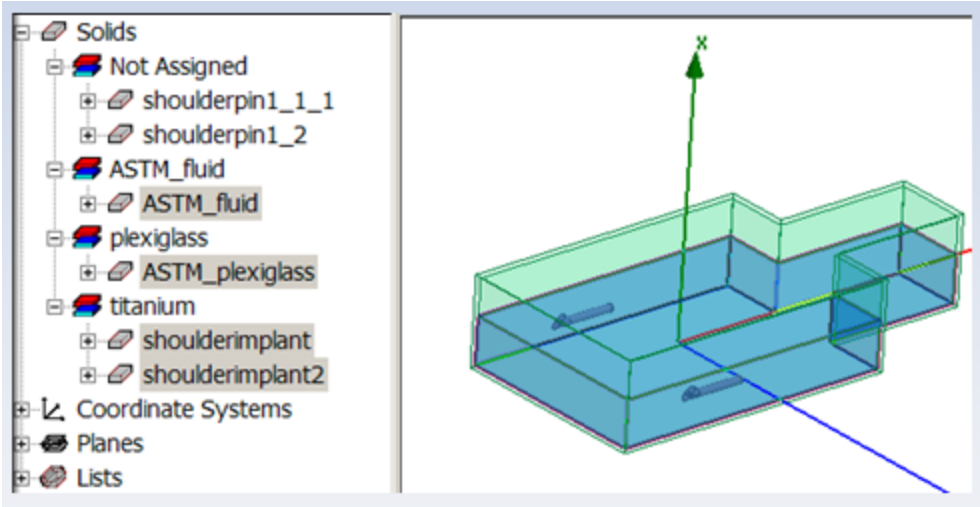
- DX analysis
- WB Update Project
- WB Update All Design Points
    - Parametric analysis with DSO
    - Animation
    - Geometry
    - Field plots



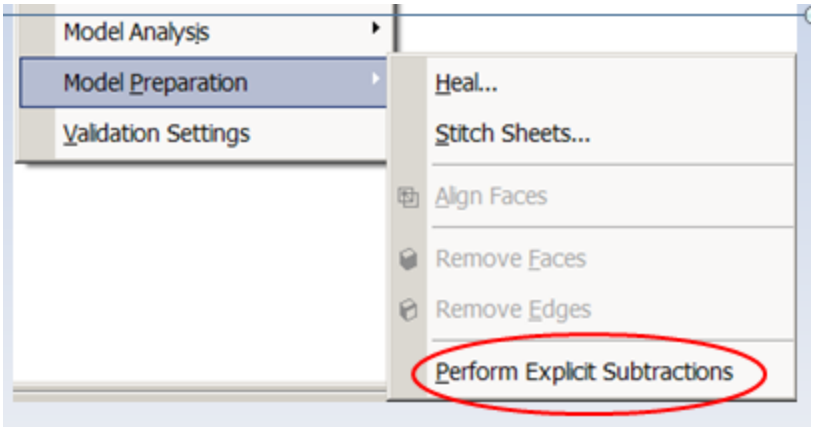
Ansys Mechanical does not do implicit subtraction. Ansys Electromagnetics products can do explicit subtractions before sending geometry to Ansys using **Modeler > Model Preparation > Perform Explicit Subtraction**.

For example, consider the following model.

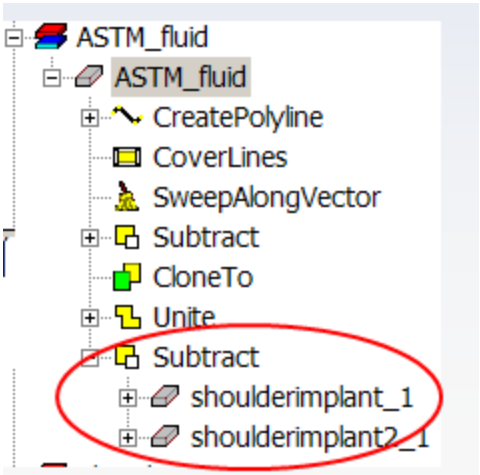
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Perform Explicit Subtractions can be performed.

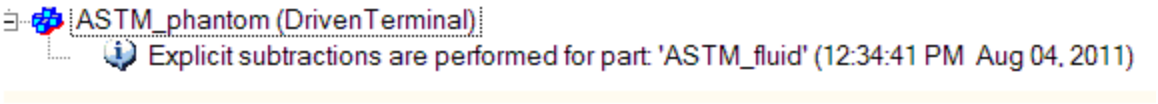


Results appear in the History tree, as shown:



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.



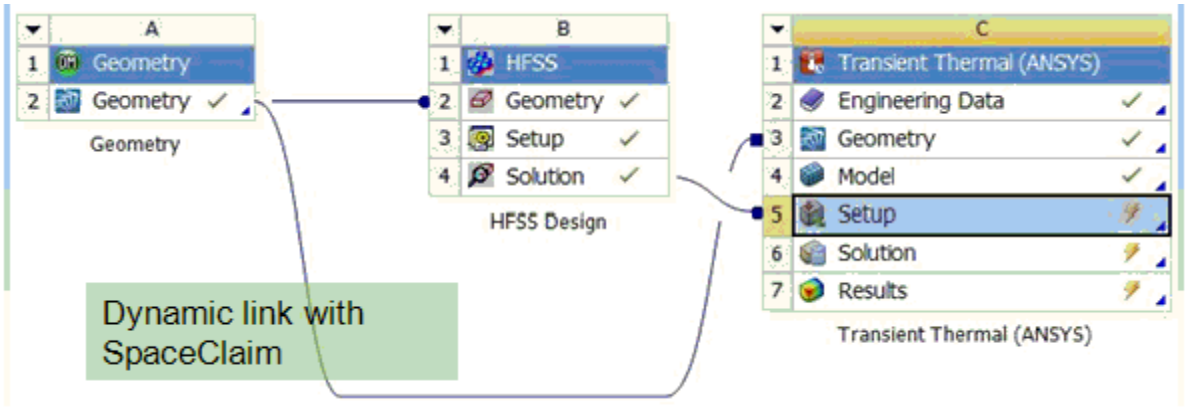
Me

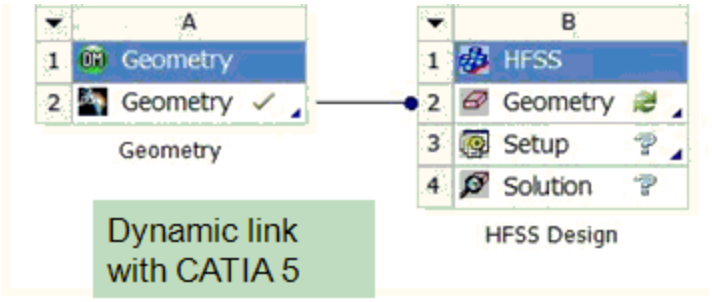
CAD Integration and Geometry Sharing

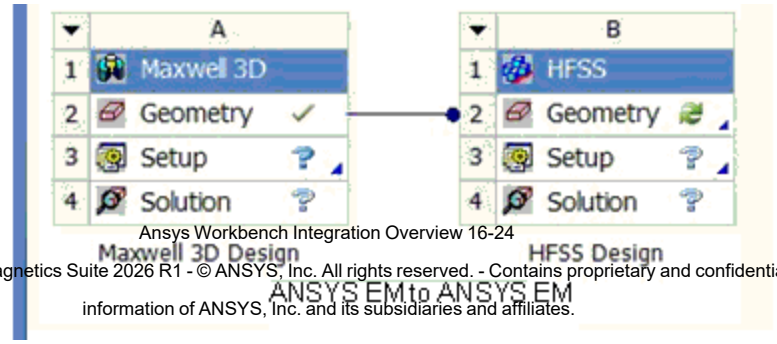
A CAD model comes into Ansys Electromagnetics as a User Defined Model (UDM).

The inputs to Ansys Electromagnetics from CAD are

- Geometry/Topology with persistent IDs
- CAD parameters
- Material assignment
- Attributes, such as name and color

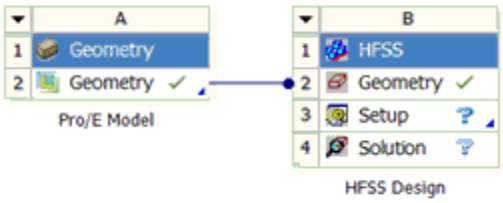




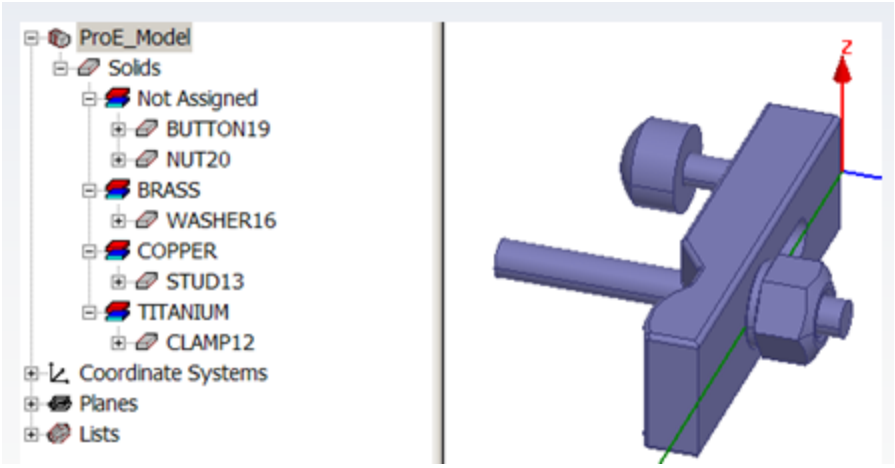


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in Workbench, a Pro/E Model can be linked to Ansys:



The geometry can then be viewed as a UDM.



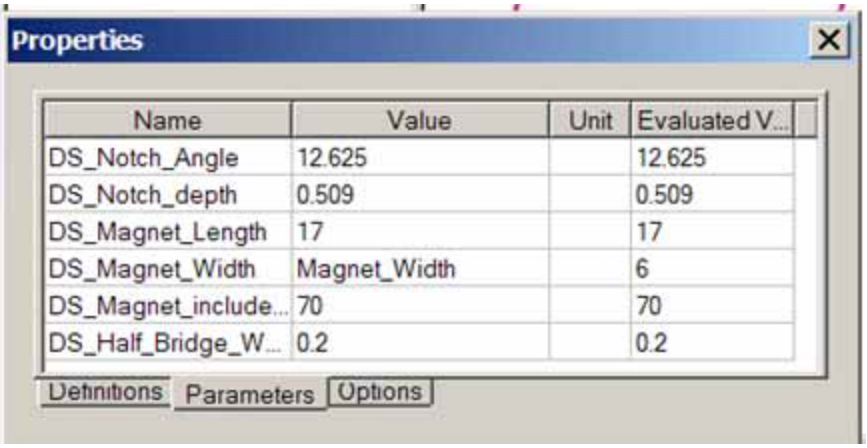
The CAD or WB model parameters appear in the Workbench:

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	

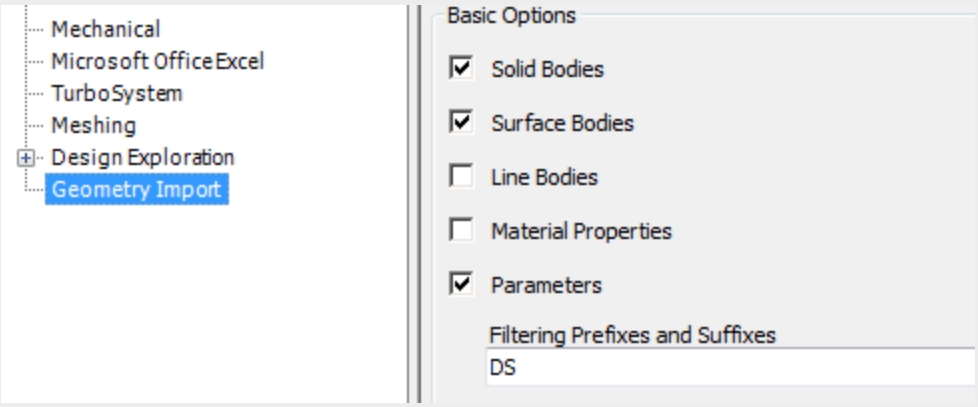
Through the Ansys Electromagnetics CAD integration, the linked UDM includes the same parameters.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



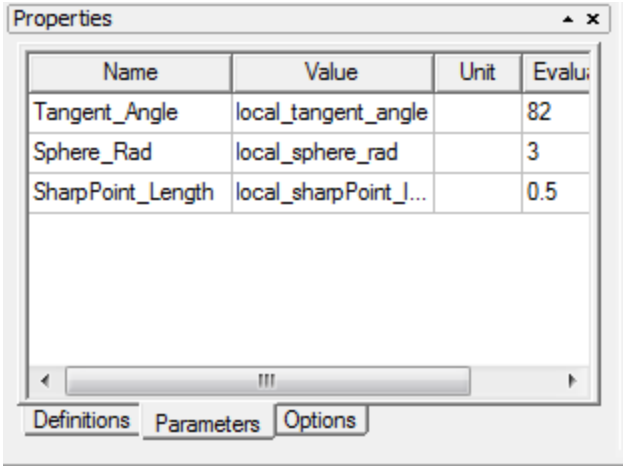
Note:

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.

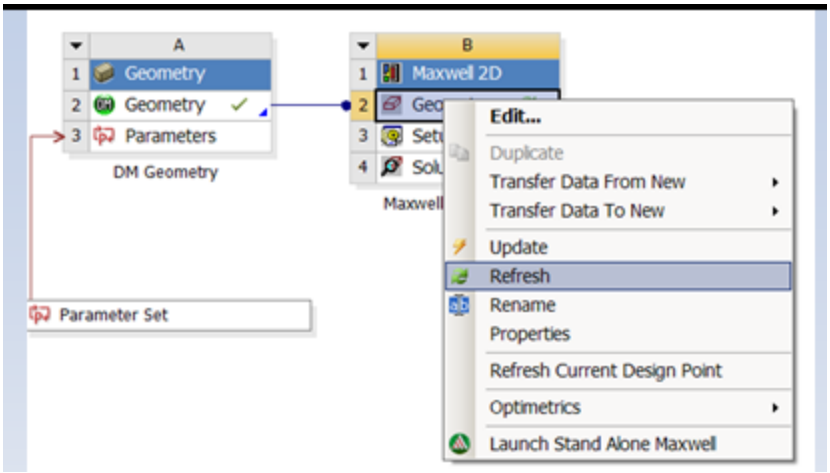
Table of Schematic D2: Design of Experiments (Custom)

	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.

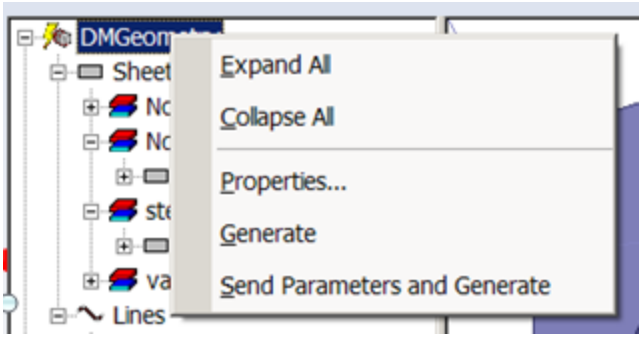


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

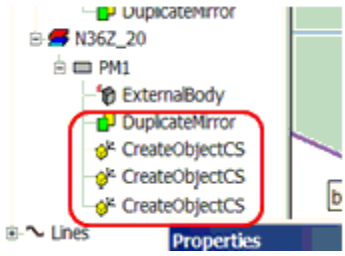
If you edit UDM (CAD) parameters in the Ansys Electromagnetics modeler window you can run the **Send Parameters and Generate** command.



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**.

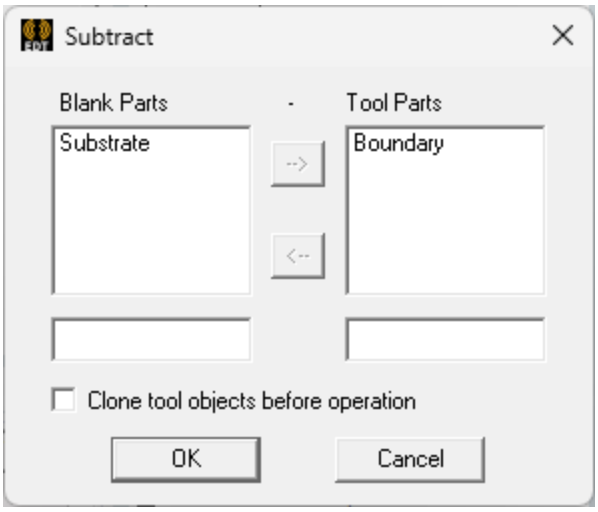


The following operations are not allowed:

- Non-history tree operations like Heal or defeature.
- Operations that use UDM parts as tools, such as Sweep

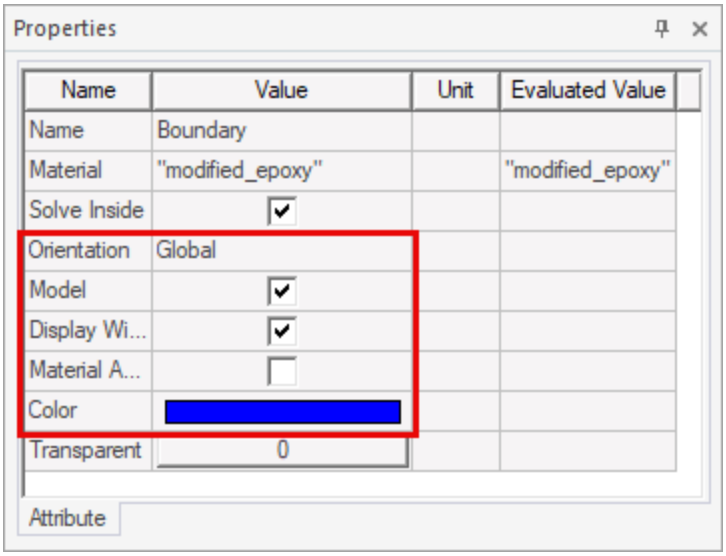
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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
- Part orientation
- Color
- Display Wireframe



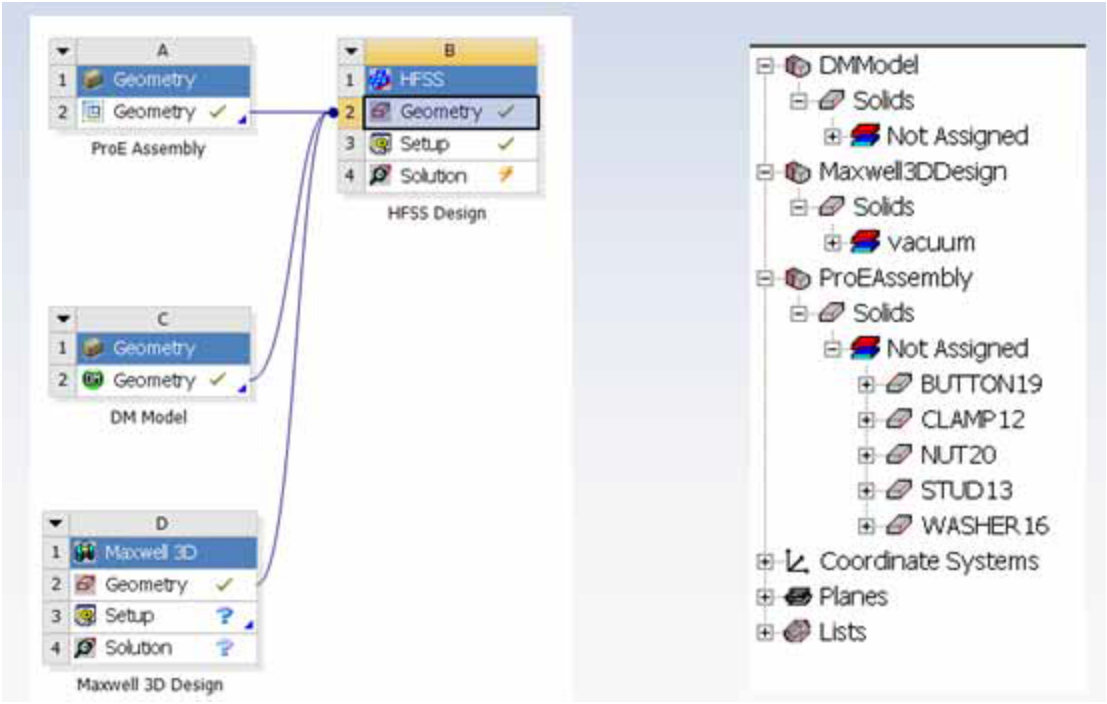
It is not possible to delete individual parts of UDM.

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.

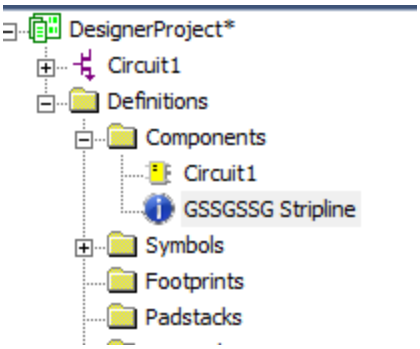


Creating Dynamic Links to EDT Designs

If you have a Circuit Design and one or more HFSS, 2D Extractor, Q3D, Icepak, or IcepakFEA designs open in a Workbench project, you can create dynamic links to those designs.

Update the Circuit Design to open the Electronics Desktop. In the menu bar, select **Workbench** > **Update Dynamic Link Components**. This command will create components in the project tree for each design in the Workbench project.

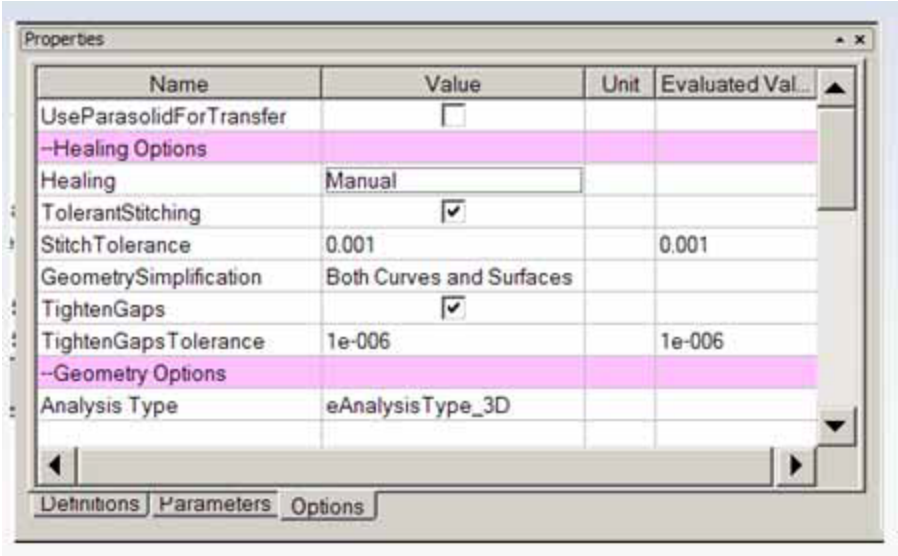
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that in this example, a GSSGSSG Stripline component was created corresponding to the GSSGSSG Stripline design in WorkBench. Once created, you can drag the component into a schematic to place it.

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.

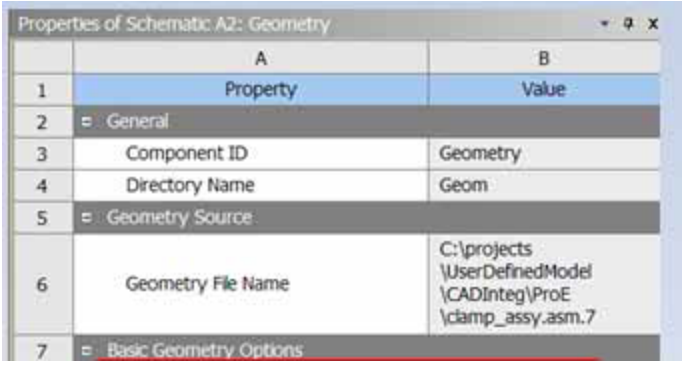


Healing options are None, Auto and Manual. By default, healing is off (None) and should be turned on only if required.

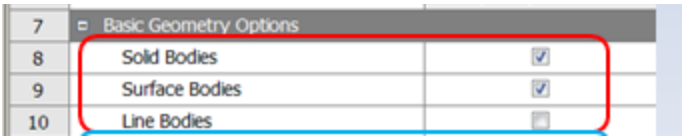
Important Geometry Options for CAD Integration

Select a Geometry Cell in Workbench to see options in the Properties window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

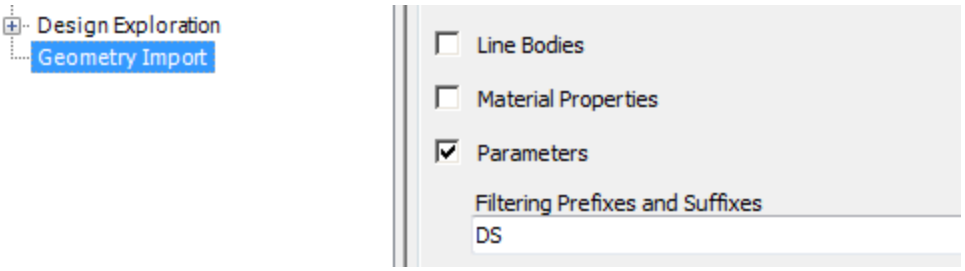


Control dimension of bodies coming from CAD



Make sure **Parameters** is checked and that the parameter key (filter) is appropriate for CAD parameters.

The default parameter key filtering means that only parameters whose names start with DS will come through. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



The **Attribute Key** should be empty or "Color" to bring in CAD colors.

Material Properties must be checked to bring in the material assignment.

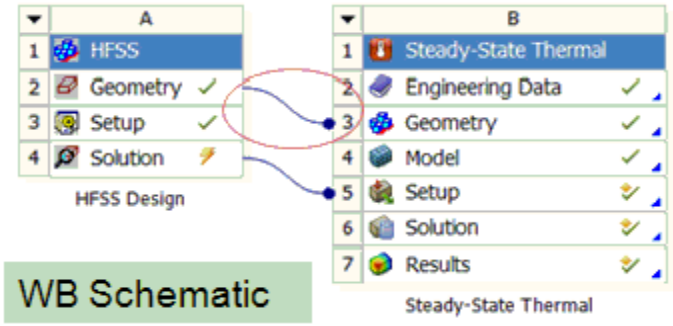
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Mixed Import Resolution** option is used to resolve parts with mixed dimension (typically from Pro/E)

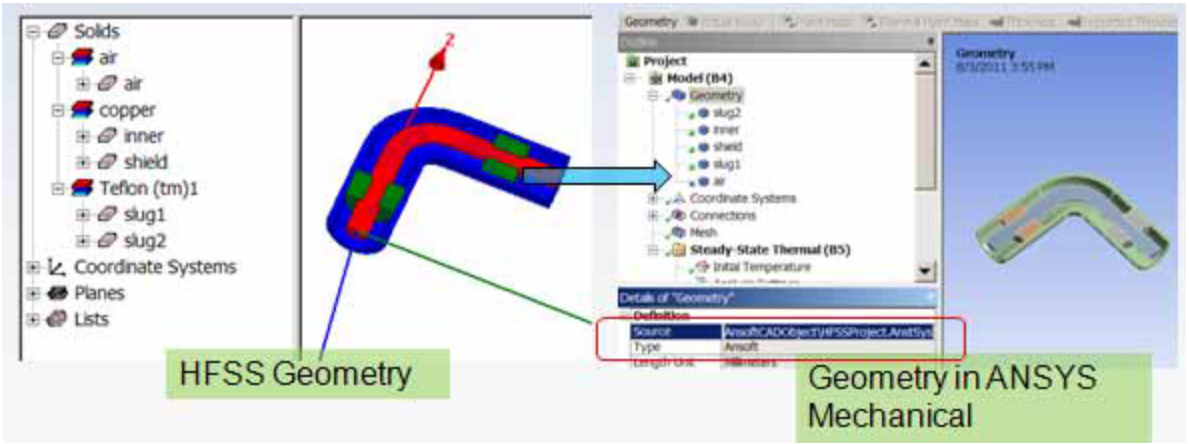
Refer to Ansys Help for details.

Ansys EM to Ansys Geometry Transfer

Ansys Electromagnetics CAD Integration transfers model information based on Workbench links.



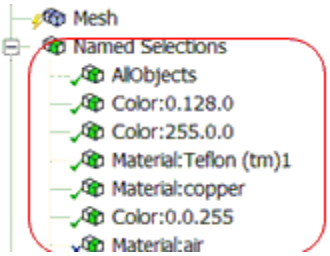
The following figure shows how the information is transferred between simulators.



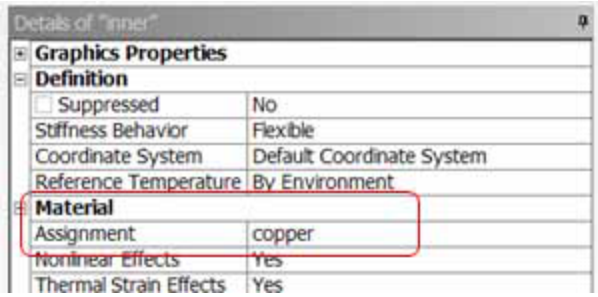
Information transferred includes

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Geometry
- Ansys Electromagnetics lists and material assignment as Named Selection



- Material assignment



The CAD Integration geometry link is dynamic (you can get updated geometry from Ansys Electromagnetics) and associative (IDs persist between Ansys Electromagnetics model and Ansys model during model refresh).

Boundary conditions in Ansys are preserved. Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.

Material Assignment Transfer

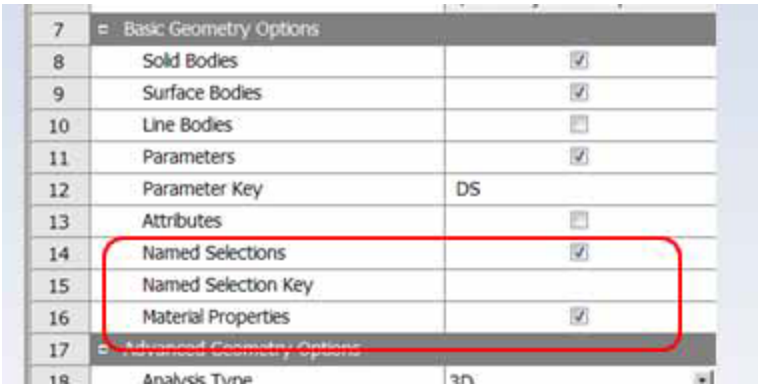
There are two prerequisites for material assignment transfer:

- 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."

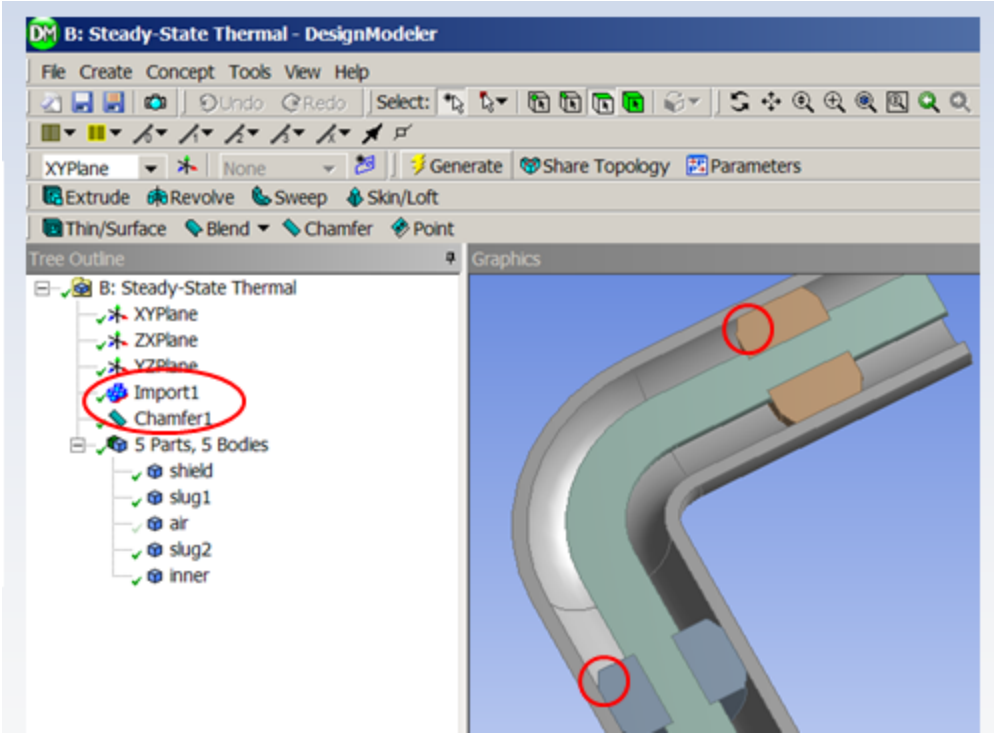
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Geometry Transfer through Ansys DesignModeler (DM)

It is possible to edit geometry in Ansys DesignModeler (DM) before consuming it in HFSS 3D Layout. 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.

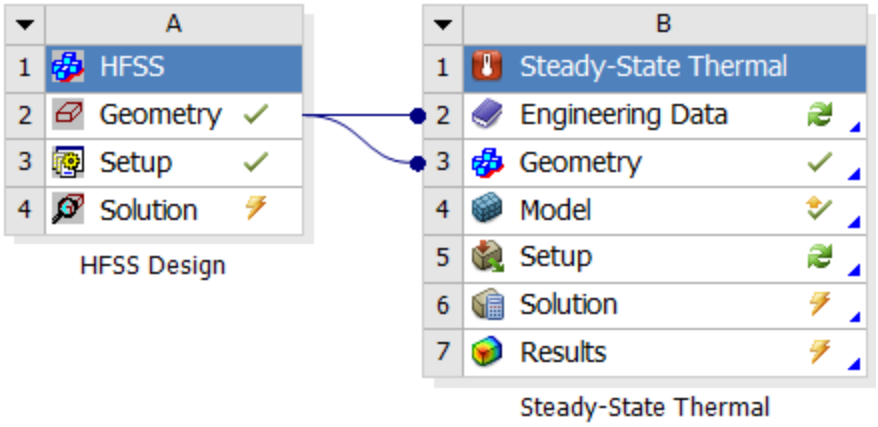


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Workbench Material Data Transfer

A link can be created from an Ansys Electromagnetics Desktop system (e.g., HFSS) geometry cell to a downstream Engineering Data (ED) cell. When the downstream ED cell is refreshed, it updates with materials (names and properties) used by the upstream project. This ensures that the geometry-to-geometry link connecting the same two systems, which specifies the material assignments for various parts, refers to the correct materials and their properties.

Detailed Behavior



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

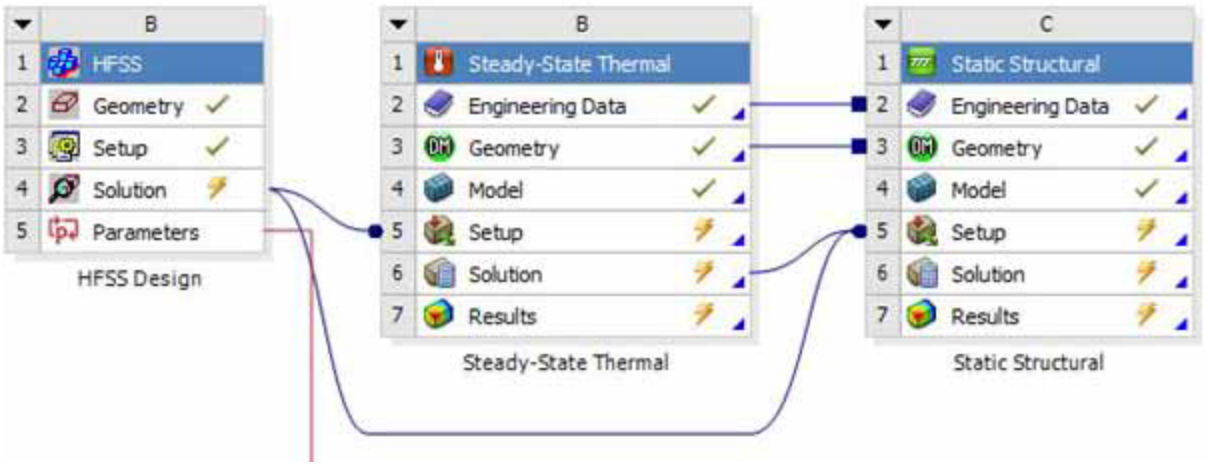
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Stress Feedback to HFSS using Workbench

Including stress feedback to HFSS involves coupling an HFSS system with both thermal and stress systems. The thermal system serves as an upstream system to the stress system and simulates the effect of heat induced displacement. And through the Workbench link between the Solution cell of HFSS and the setup cell of Mechanical, the displacement can feedback into HFSS. The HFSS project can be Driven or Eigenmode.



You use a dialog to select one or more objects (geometry) in HFSS upon which displacement data from Mechanical applies. This feature lets you avoid applying displacement to the whole model where only certain parts are sensitive to stress. You must select the same objects in the thermal and stress system to apply the respective imported loads.

The results of the process include deformation feedback applied to selected objects. The Solution Profile includes a report of Solve with thermal/displacement feedback, including information Maximum Delta T and Maximum Delta Displacement. You can create field plots with based on Displacement and Mag_Displacement, and you can scale the deformation displayed. The Fields Calculator also includes these input quantities.

Interface Changes for Stress Feedback from Mechanical to HFSS

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Process Flow for Stress Feedback to HFSS

Interface Changes for Stress Feedback from Ansys Mechanical to HFSS

When used through the Workbench for stress feedback projects, the HFSS interface contains some commands and information that do not appear in standalone HFSS.

Deformation of Objects dialog

Solution Profile information for Solve with Displacement Feedback

Plot Fields Menu with Displacement and Mag_Displacement

Deformation Scale tab for Modify Plot Attributes

Revert to Zero Displacement command under the **Solve Setup** menus

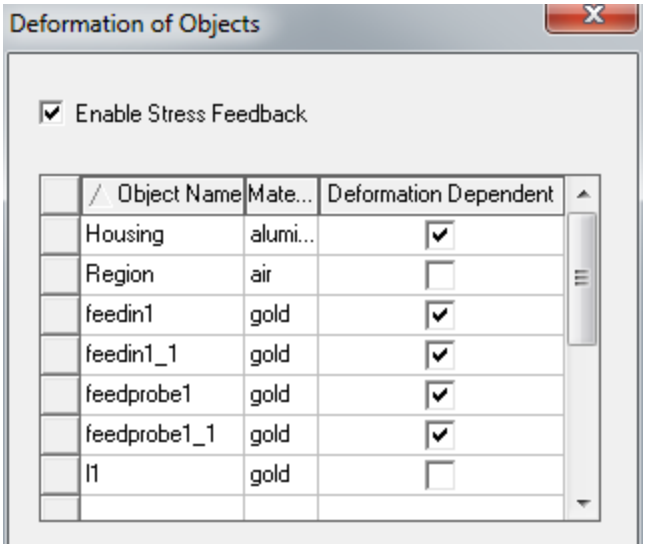
Fields Calculator Field Quantities for Stress Feedback Projects

Plot Fields Context Menu for Stress Feedback Projects

Modify Plot Attributes Dialog for Stress Feedback Projects

Deformation of Objects Dialog

When working with HFSS through Ansys Workbench, the HFSS menu for Driven and Eigenmode projects includes a Deformation of Objects command that opens this dialog for specifying which objects shall be deformation dependent.



The Enable Stress Feedback check box enables object selection. All selections are disabled when this is unchecked. Changing the setting invalidates any existing solutions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

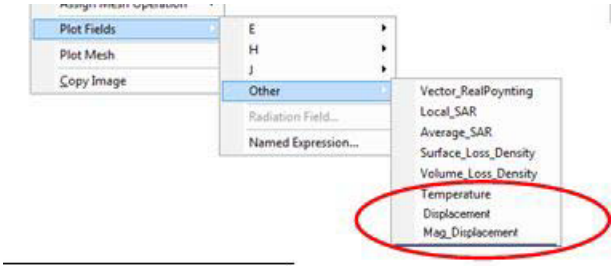
The objects listed do not include objects without materials assigned, non-model objects, or non-3D objects. Objects assigned materials such as air or vacuum are not deformed and don't apply to stress feedback.

Solution Profile with Displacement Feedback

The Solution Profile includes a report of Solve with thermal/displacement feedback, including information Maximum Delta T and Maximum Delta Displacement.

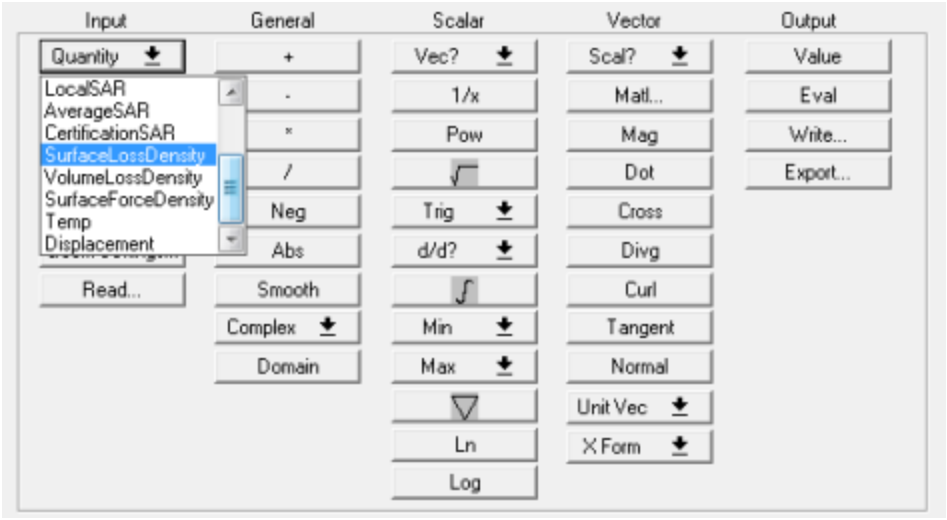
Plot Fields Menu with Displacement and Mag_Displacement

The Plot Fields menu includes entries for displacement and mag_displacement.



Fields Calculator Field Quantities for Stress Feedback Projects

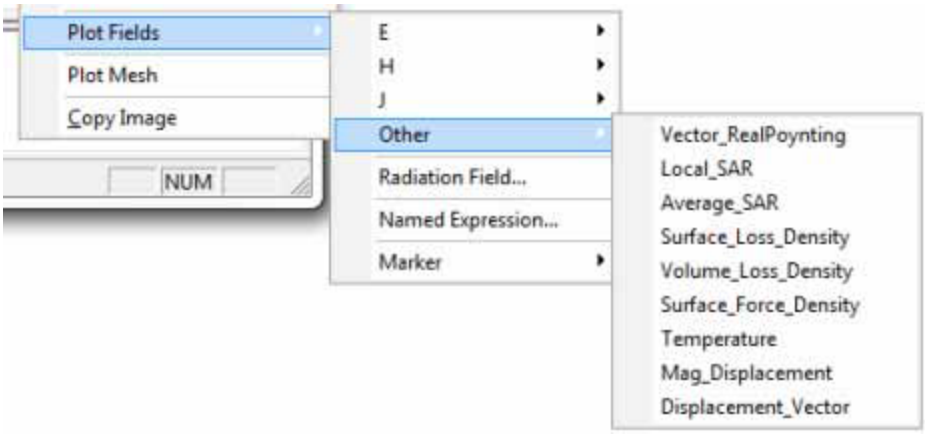
The Quantity menu for the Fields Calculator lists Displacement for Stress Feedback projects.



Plot Fields Context Menu for Stress Feedback Projects

The **Plot Fields>Other** menu includes Displacement_vector and Mag_Displacement, a scalar. A field plot remains empty until the displacement data applies to the HFSS solution.

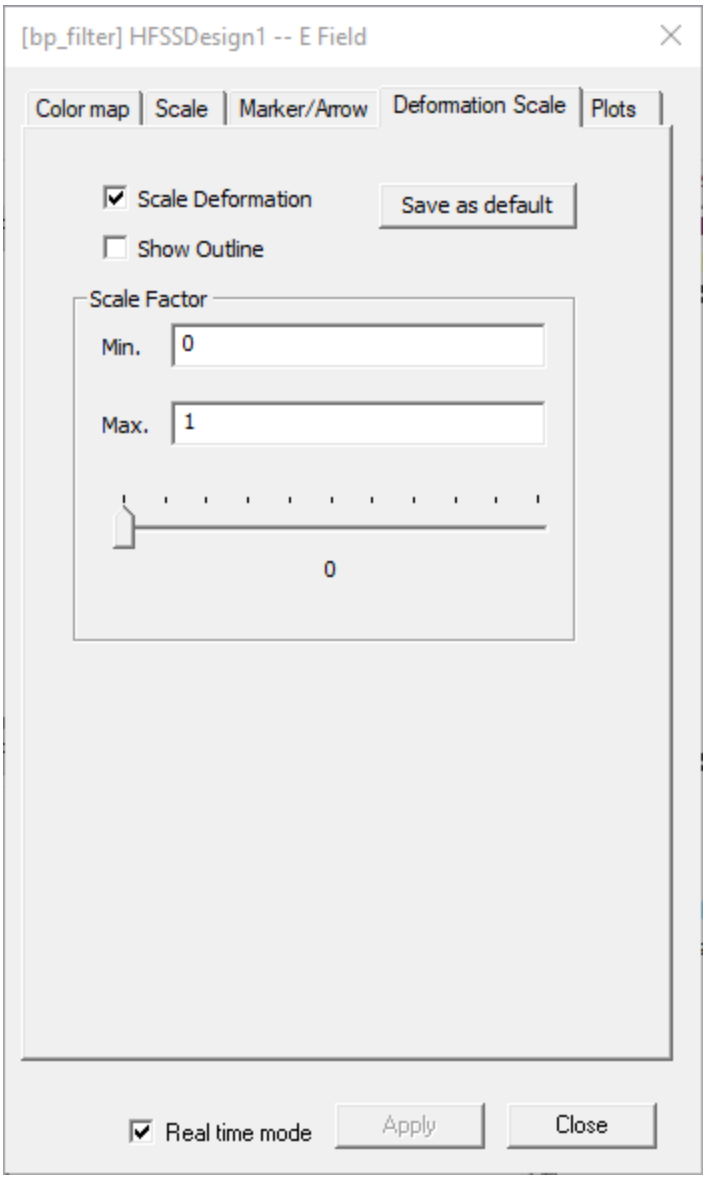
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Modify Plot Attributes Dialog for Stress Feedback Projects

When an HFSS plot includes deformation, the **HFSS>Fields>Modify Plot Attributes** dialog includes a tab for scaling the deformation. If you check Scale Deformation, you can also select Show Outline.

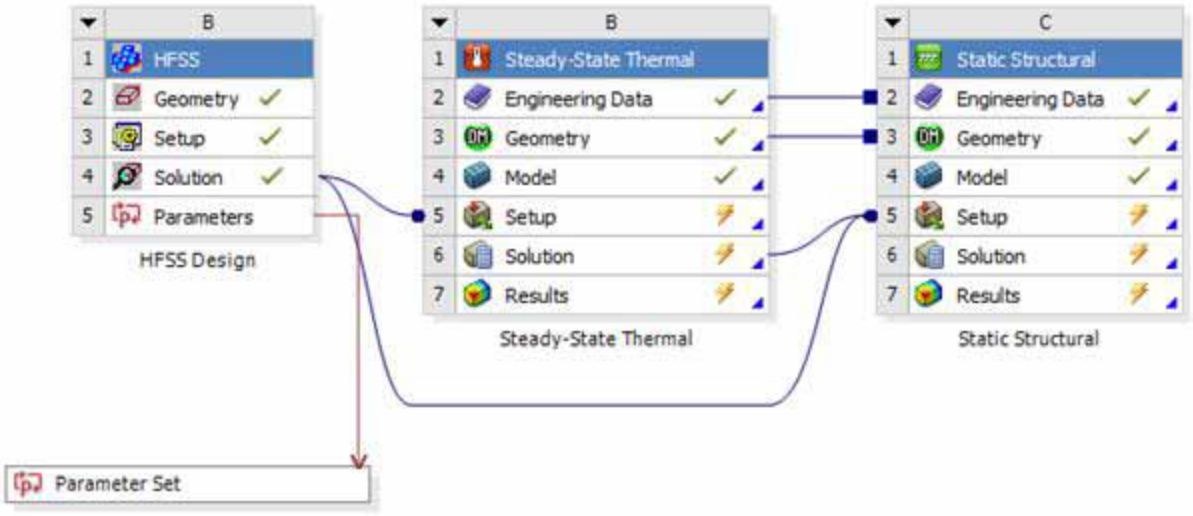
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



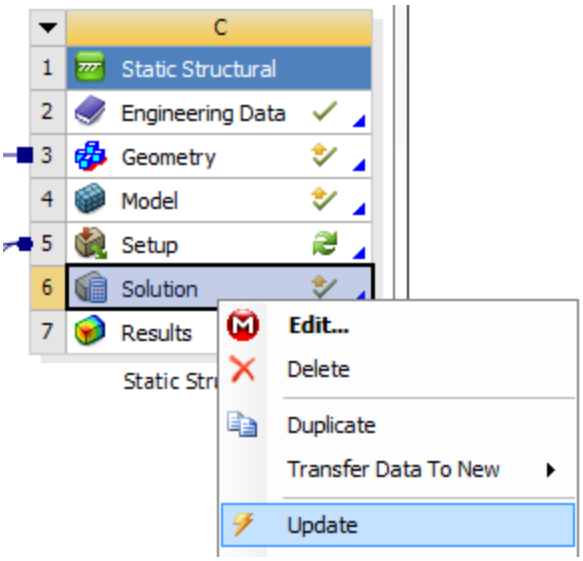
Process Flow for HFSS Workbench for Stress Feedback

to set up the coupling, drag the "Solution" cell of the HFSS system and drop it at the "Setup" cell of a Thermal and Stress system. You also need to also couple the Thermal and Stress system to capture the effect of thermal force. The following image illustrates a coupling setup where both the Thermal and Stress system are ready to be "Updated". The HFSS adaptive solution has converged in 3 passes. The HFSS system has both thermal and stress feedback enabled.

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 the Solution cell of Stress and select "Update"

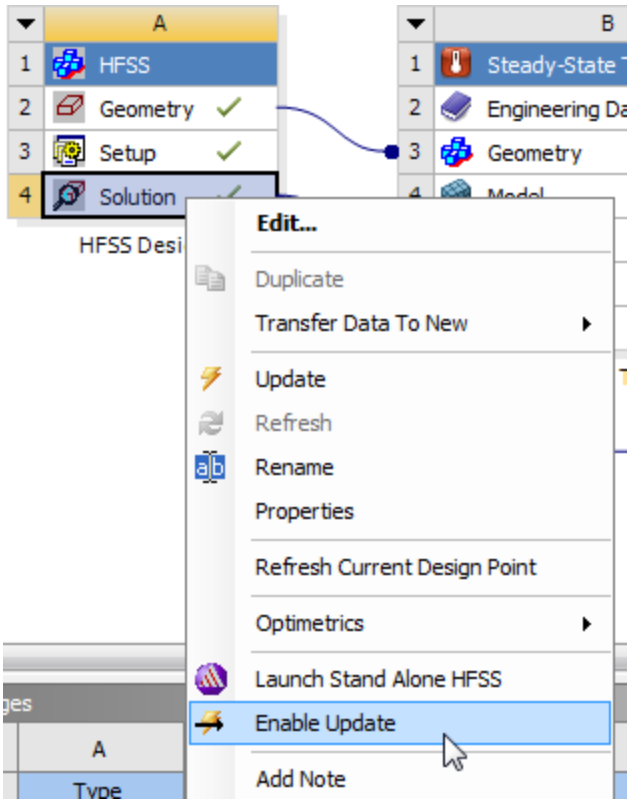


   a. The Setup of Thermal will be "Updated" with em loss from HFSS
   b. The Solution of Thermal will be "Updated" and temperature will be exported to HFSS

   c. The Setup of Stress will be "Updated" with thermal force from Thermal and force density (just zeros) from HFSS
   d. Displacement will be exported to HFSS after Stress finishes simulation

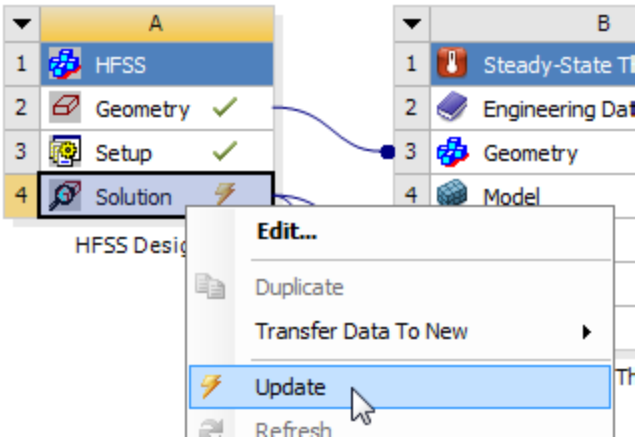
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Right-click the Solution cell of HFSS and select "Enable Update"



The Solution cell icon changes from a green checkmark to the Update lighting icon.

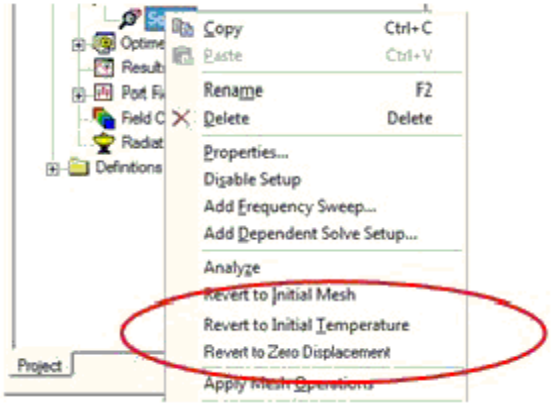
3. Right-click the Solution cell of HFSS and select "Update"



   a. HFSS re-simulate the 3rd pass with its mesh and the temperature/displacement feedback
   b. The profile will show information about the feedback

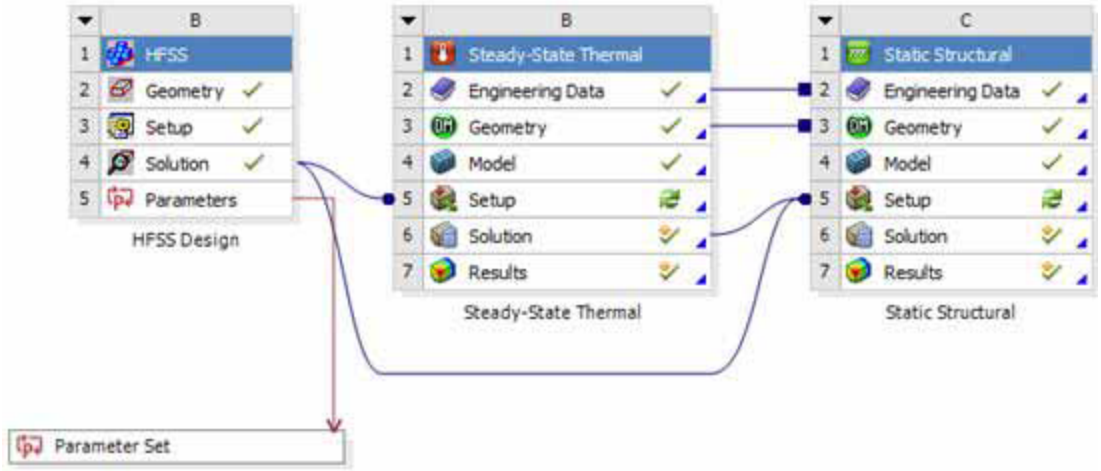
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Both "Revert" menus will be available at the right mouse click menu of the solve setup item.



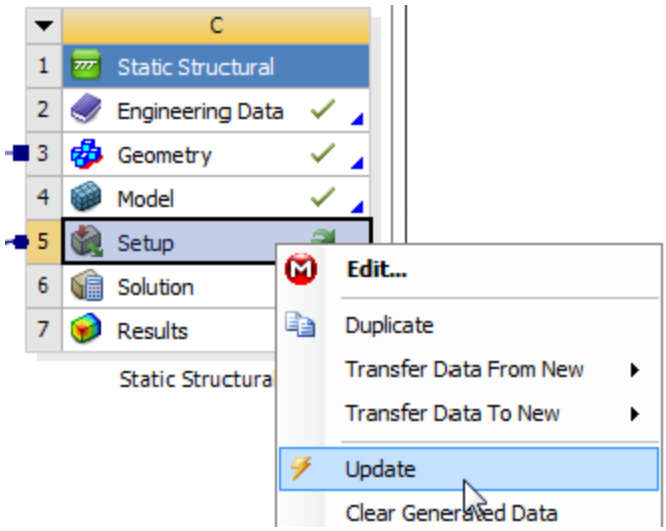
Manual Iteration After First Iteration Completes

The following image illustrates the WB schematic after the previous situation, after the 1st iteration has completed. Notice the changed icons for the status for Setup for Thermal and Structural.

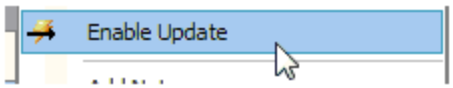


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

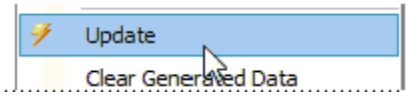
1. Right-click the Solution cell of Stress and select "Update"



   a. The Setup of Thermal will be "Refreshed" and "Updated" with EM loss from HFSS
   b. The Solution of Thermal will be "Updated" and temperature will be exported to HFSS
   c. The Setup of Stress will be "Refreshed" and "Updated" with thermal force from Thermal and force density from HFSS
   d. Displacement will be exported to HFSS after Stress finishes simulation
2. Right-click the Solution cell of HFSS and select "Enable Update"



3. Right-click the Solution cell of HFSS and select "Update"



   a. HFSS re-simulates the 3rd pass with its mesh and NEW temperature/displacement feedback
   b. The profile will show information about the feedback

Note that the delta temperature and displacement is being reported in the profile

Revert the HFSS Solution For Temperature and Displacement

You can revert the temperature and displacement in HFSS solution separately.

1. Select "Revert to Initial Temperature" in HFSS.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A warning message notifies you about the invalidation of solution

2. Right-click HFSS solve setup and select "Analyze"

HFSS re-simulates the 3rd pass with its mesh and displacement that was previously exported from Mechanical, but without temperature.

3. Select "Revert to Zero Displacement" in HFSS, and followed by "Analyze"

HFSS re-simulates the 3rd pass with its mesh without neither temperature nor displacement

Only Stress Feedback

The coupling setup is the same as in Use Case 1, with either one of the following differences

- The HFSS system is not enabled for thermal feedback. Since HFSS is not enabled to support feedback, the "Export Result" properties will be missing in the thermal system's property window.
- The HFSS system is enabled for thermal feedback, but users select not to "Export Result" in the thermal system. This is to disable the "automatic" feedback after Mechanical finishes its simulation.

1. Right-click the Solution cell of Stress and select "Update"
    a. The Setup of Thermal will be "Updated" with em loss from HFSS
    b. The Solution of Thermal will be "Updated"
    c. The Setup of Stress will be "Updated" with thermal force from Thermal and force density from HFSS
    d. Displacement will be exported to HFSS after Stress finishes simulation
2. Right-click the Solution cell of HFSS and select "Enable Update"
3. Right-click the Solution cell of HFSS and select "Update"
    a. HFSS re-simulate the 3rd pass with its mesh and the displacement feedback
    b. The profile will show information about the feedback and "Revert to Zero Displacement" will be enabled at the RMC menu of the solve setup item

Feedback Iterator

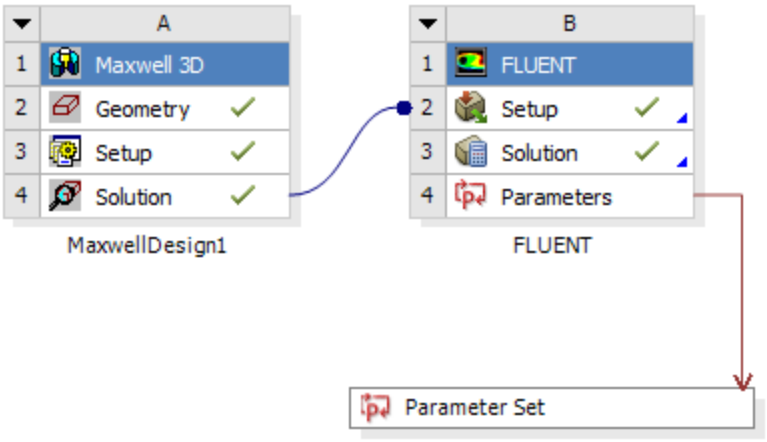
Background Information

Prior to the introduction of the Feedback Iterator, Ansys Workbench supported a two-way loose-coupling protocol with Ansys Electromagnetics products.

System Coupling already uses the word coupling to mean low-level solver coupling. Existing electromagnetics product coupling is loose compared to system coupling and is limited to file transfers at the end of a complete solve in a standalone system/product. No communication occurs during a solve. The coupling is two-way.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The one-way portion (Upstream source component to Downstream target component) is handled via existing Workbench data/transfer connection mechanism.

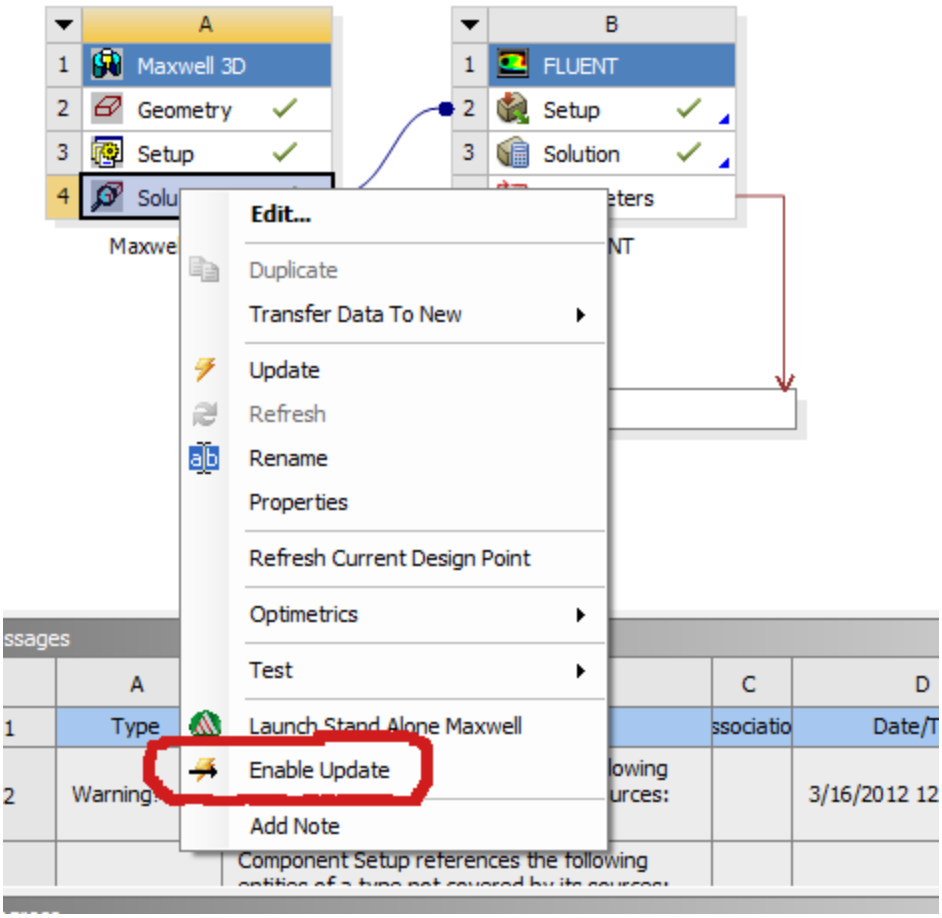


The round trip is handled by a separate protocol, agreed upon by the participating systems, whereby the downstream system exports a set of files to a location specified by the upstream system via its one-way transfer. This exported data is then incorporated by the upstream system in its next update.

To run the next coupled solve iteration, invoke the **Enable Update** operation, as shown below, which updates all systems involved.

These steps (**Enable Update**, **Update Project**) are continued as long as needed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



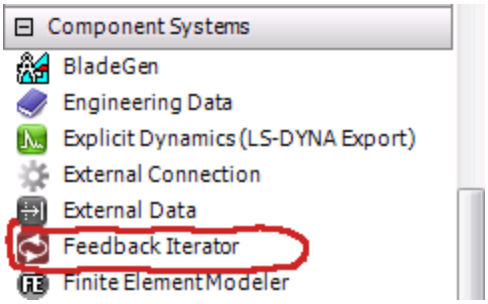
The Feedback Iterator System

Ansys Electromagnetics Suite provides the Feedback Iterator system for automating the manual steps needed for driving a feedback utilizing system-pair to convergence. In addition to automating feedback incorporation over a user-specified number of iterations, the Feedback Iterator also allows control over the number of iterations (iteration termination criteria), target temperature, and displacement convergence criteria.

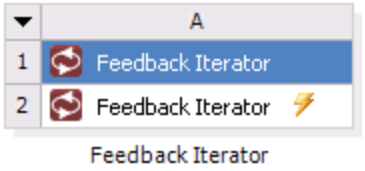
> **Note:** The Feedback Iterator is required if there are temperature-dependent or stress-dependent material properties.

The **Feedback Iterator** appears in the Workbench Toolbox user interface under **Component Systems**.

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 drag and drop an instance of the Feedback Iterator into a Workbench schematic, the default system name is Feedback Iterator:

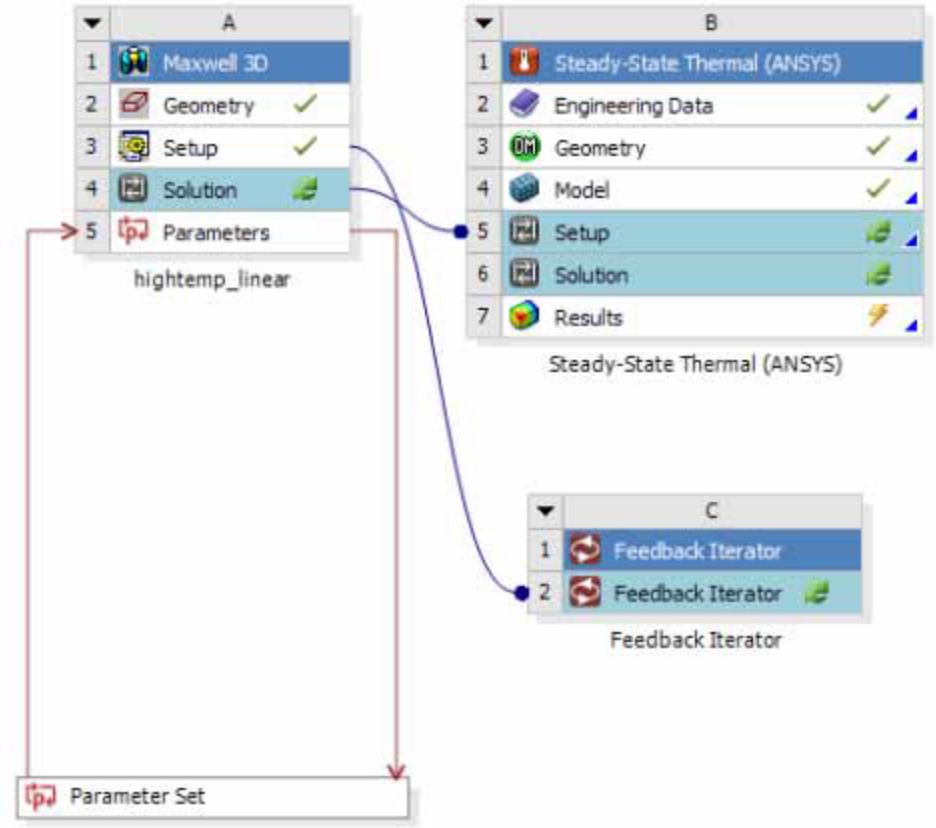


Feedback Iterator in Use

The Feedback Iterator system attaches to the Workbench schematic like any other system.

In the following example, a Maxwell system provides heat loss data to a Steady-State Thermal system. Dropping the Maxwell setup cell onto the Feedback Iterator component results in the links 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 Feedback Iterator add-in functions the same way with all systems that participate in the Electromagnetics product two-way loose coupling (Maxwell and HFSS for sources and Fluent, Thermal, and Structural for targets).

Feedback Iterator Component Properties

The Feedback Iterator's primary role is to control iterative solves and its properties target this end.

For Maxwell and HFSS, the Feedback Iterator properties are

- **Iterations Completed** – a read-only property that displays the number of iterations completed.
- **Max Iterations** – sets the maximum number of iterative solve loops to perform before terminating iterations in case the *Target Delta Temperature %* or *Target Delta Displacement %* is not achieved. The default value is 100.
- **Target Delta Temperature %** – specifies the maximum *Target Delta Temperature %* that signifies convergence. The default value is 5%, and the value must be 0.01 or greater.

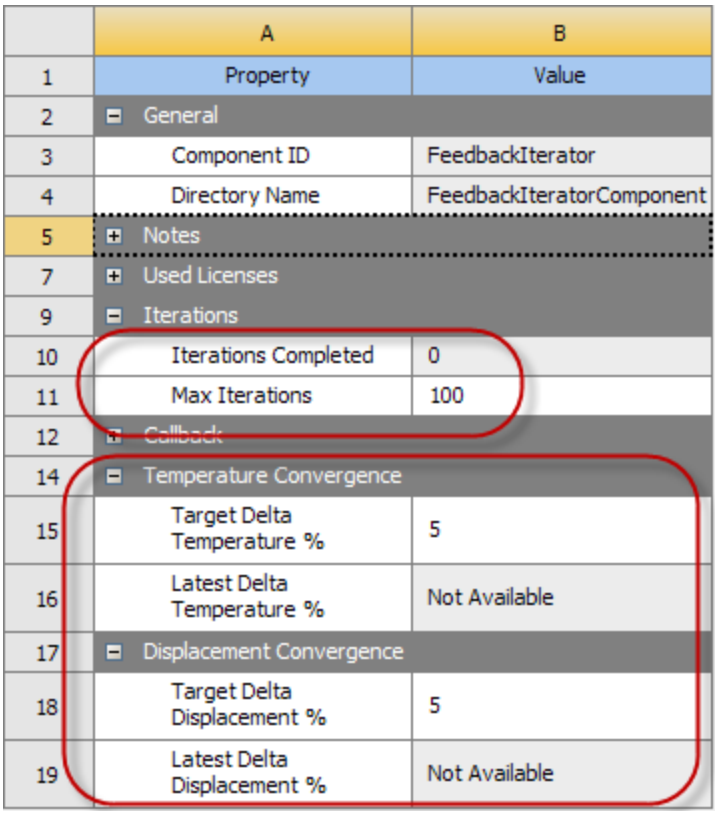
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

• The delta temperature error is calculated in Kelvin.

• The Maxwell and HFSS design **Profile** tab displays absolute/relative delta data, while the delta temperate in Workbench means the relative delta (expressed as %).

- **Target Delta Displacement %** – specifies the maximum *Target Delta Displacement %* that signifies convergence. The default value is 5%, and the value must be 0.01 or greater.



	A	B
1	Property	Value
2	⊟ General	
3	Component ID	FeedbackIterator
4	Directory Name	FeedbackIteratorComponent
5	⊞ Notes	
7	⊞ Used Licenses	
9	⊟ Iterations	
10	Iterations Completed	0
11	Max Iterations	100
12	⊞ Callback	
14	⊟ Temperature Convergence	
15	Target Delta Temperature %	5
16	Latest Delta Temperature %	Not Available
17	⊟ Displacement Convergence	
18	Target Delta Displacement %	5
19	Latest Delta Displacement %	Not Available

The Feedback Iterator properties also display the most recently achieved *Latest Delta Temperature %* and *Latest Delta Displacement %* values, allowing the user to manually abort when satisfied with the achieved deltas.

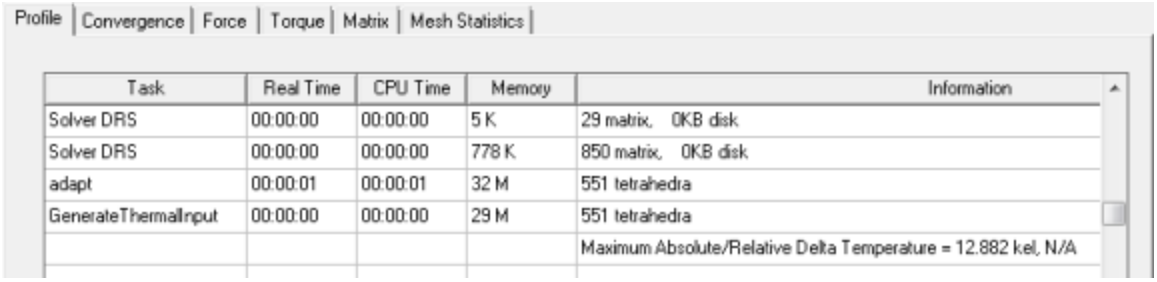
Note:

Both the Temperature and Displacement Convergence properties sections are always shown irrespective of which feedback types are enabled for the project.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Running a two-way feedback simulation from the Workbench application while keeping the Maxwell or HFSS **Profile** tab open shows that the 3D solver keeps track of two feedback related quantities:

- Maximum Absolute/Relative Delta Temperature (if temperature feedback is enabled)
- Maximum Absolute/Relative Delta Displacement (if displacement feedback is enabled)



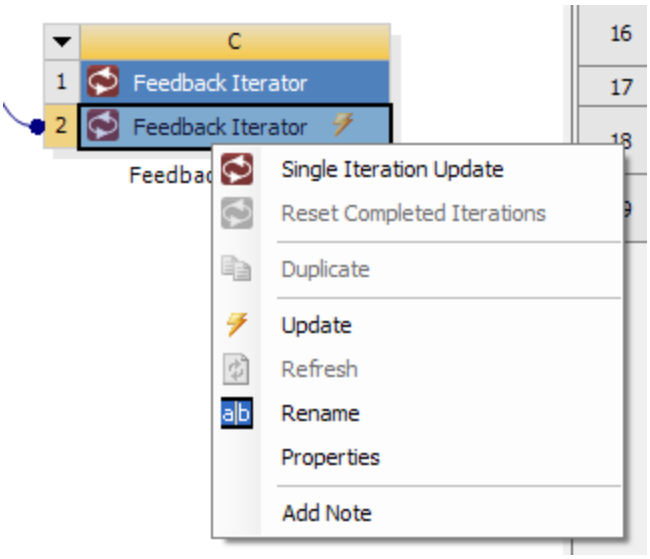
These signify the maximum difference on the solution mesh of the quantity in question from one feedback iteration to the next (thus the value is only available from the second iteration onward). This data forms the basis for the convergence controls in the Feedback Iterator. These controls allow the user to base the termination of feedback iterations on a target max delta T and/or max delta displacement. The target max delta forms the convergence condition.

Feedback Iterator User Interface Operations

The **Single Iteration Update** action allows you to run a single iteration worth of updates, as follows:

1. Updates the Electromagnetics product system, which will incorporate any previous feedback from the downstream system.
2. Updates the downstream system.
3. All coupled systems (Electromagnetics product, Electromagnetics product Downstream, and Iterator) are in the UpToDate state.
4. Iterator's “Iterations Completed” property increments by 1.

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 the Feedback Iterator component and selecting **Update** allows you to run iterations automatically. Iterations continue to run either until convergence criteria are met, or until the Max Iterations value is reached.

Callback Interface

The callback interface allows you to react to each step of the iteration process as implemented by the Feedback Iterator component. This is useful for implementing special iterations or solving transient simulations with a limited scope.

There are four main API features provided:

- Callback functions, which the Feedback Iterator calls at various points in each iteration
- Utility functions, which the Callback functions can use to extra properties of the containers being processed
- Output functions, which allow script debugging and supply additional messages
- Limited state management, which allows the script to store and retrieve state across callback functions and iterations

Callback and State API

State is managed in a limited fashion. All the API methods take a final dictionary argument. This dictionary is limited to using **string keys** and **number or string values**. Within this limitation, add new keys, read old keys, clear the dictionary, etc., and it will be persisted across functions calls and even across Updates. When the **Iterations Completed** property is **rest**, the dictionary is also cleared out.

It is advisable for the callback script to initialize the dictionary at Iteration 1.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All the API methods use a subset of the following arguments:

`iterationNumber`	An integer representing the current iteration. This always starts from 1.
`ContainerList`	A Python list of containers (DataContainerReference). This is the entire list of the coupled containers managed by the Feedback Iterator. You typically loop through them and using the utility methods listed below, identities them.
`Container`	A single container that is being processed. This is a DataContainerReference.
`State`	A read/write Python dictionary that is used to maintain state across function calls and iterations.

Only return values from `BeforeIterationEx` are processed. Returns from all other functions are discarded.

1. `BeforeIteration(IterationNumber, ContainerList, State)`: This method is called before each iteration. Ideally used to initialize the state dictionary, open editors as required, or initialize setups as needed for each iteration.
2. `BeforeIterationEx(IterationNumber, ContainerList, State)`: Similar to the `BeforeIteration` method except that this allows you to control the number of iterations via the return value.
    - Return: `"more"` to request one more iteration
    - Return `"last"` to indicate that this is the last iteration
    - Any other return (including none) will be treated as a return of `"last"` and terminate iterations.
3. `AfterIteration(IterationNumber, ContainerList, State)`: This method is called after each iteration. This can be used to copy result files over, check results, implement any possible convergence calculations, logging of results, *etc*.
4. `BeforeContainerRefresh(IterationNumber, Container, ContainerList, State)`: Is called before each of the coupled containers is refreshed. The `Container` argument represents the container about to be refreshed.
5. `AfterContainerRefresh(IterationNumber, Container, ContainerList, State)`: Is called after each of the coupled containers is refreshed. The `Container` argument represents the container just refreshed.
6. `BeforeContainerUpdate(IterationNumber, Container, ContainerList, State)`: Is called before each of the coupled containers is Update (after a refresh). The `Container` argument represents the container about to 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.

7. `AfterContainerUpdate(IterationNumber, Container, ContainerList, State)`: Is called after each of the coupled containers is updated. The `Container` argument represents the container just updated.

If the callback scripts uses other files to send commands to various containers (vb, js, apdl, python, etc.), all of those files are best saved under the user_files directory. This allows you to use the `FBGetUserFilePath(str)` to get the absolute path of the file and allows the files to be packaged with any created archive.

Utility Functions

`FBSystemForContainer` (container)	system	Given a container, returns the system it belongs to
`FBSystemDisplayName` (system)	string	Given a system, returns its display name on the schematic
`FBSystemID`(system)	string	Given a system, returns its ID. This is the same as the `UniqueDIrectory` for the system
`FBGetUserFilesPath` (relativePath)		Given a relative path located under the user_files directory, say "hello.py", this returns the absolute path of the file

Output/Debugging Functions

`FBAddInfoMessage (string)`	Adds an info message to the WB message window
`FBAddWarningMessage (string)`	Adds a warning message to the WB message window
`FBAddErrorMessage (string)`	Adds an error message to the WB message window
`FBMessageBox (string)`	Pops up a dialog box with the supplied string with an **OK** button

Example Scenarios for Feedback Iterator

This section describes several scenarios for using the Feedback Iterator:

Setting up Iteration with Feedback Iterator

Breaking Iteration Control

Starting an Iterative Update

Running a Single Iteration

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interrupting an Iterative Loop

Resuming an Interrupted Iterative Loop

Modifying the Systems Involved in Iteration (coupled clients)

Iterating to Convergence

Setting up Iteration with Feedback Iterator

> **Note:**
>
> A feedback iterator is required if there is a temperature-dependent or stress-dependent material property.

Scenario	Steps to execute	Outcome
Electromagnetics product solution provides solution data to a single downstream setup	• Drop Feedback Iterator system on the setup component of Electromagnetics product OR • Create Feedback iterator system and connect Electromagnetics product setup component to the TwoWay iterator component AND User clicks on the Iterator component and sets the desired Feedback Iterator properties.	Electromagnetics product setup component is connected to the Iterator. Electromagnetics product solution and downstream setup and solution components are coupled as clients to the iterator. The iterator component is the coupled master and the rest are coupled clients. Any coupling changes schematic visuals: • Coupled cells including coupling master are colored differently from normal cells • Coupled cell icons change to reflect the icon of the coupling master • Coupled client cells no longer display the **Update** context menu item and cannot be updated via script commands • The cell states of all coupled cells (master and clients) are synchronized to be the most pessimistic state among any of them (e.g., after an update, any modification will set all of them to the "Modified" state).
Electromagnetics product solution provides to multiple	same as above	All the downstream components (setup and solution in each downstream system) are coupled as clients. The update order in this case depends on any

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
downstream setup cells		additional data-flow connections between the systems that consume the upstream Electromagnetics product solution data.
Electromagnetics product solution cell does not have any downstream targets	same as above	No changes occur beyond what is expected when you create a new connection (no coupling). Coupling only occurs when Electromagnetics product solution cell has a downstream connection. If a downstream connection is added after the link to the iterator, coupling will be performed as described in the first row.

Breaking Iteration Control

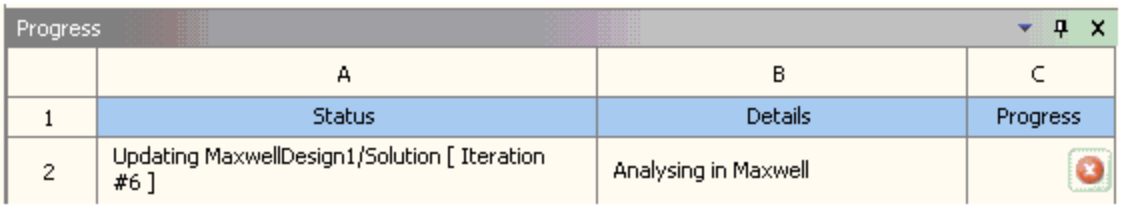
Scenario	Steps to execute	Outcome
Iterator coupled to Electromagnetics product solution cell and downstream setup/solution cells	• Break Electromagnetics product solution cell's provides link OR • Break Electromagnetics product setup link to iterator OR • Delete Electromagnetics product solution consumer system OR • Delete Iterator system	In addition to what is normally expected from the user actions, all coupled clients are decoupled (their icons and colors on UI are restored)
Electromagnetics product setup cell is connected to iterator, but Electromagnetics product solution does not provide anything. No	• Break Electromagnetics product setup link to iterator • Delete Iterator	No coupling exists in this scenario so nothing visible changes beyond what is expected from the user actions

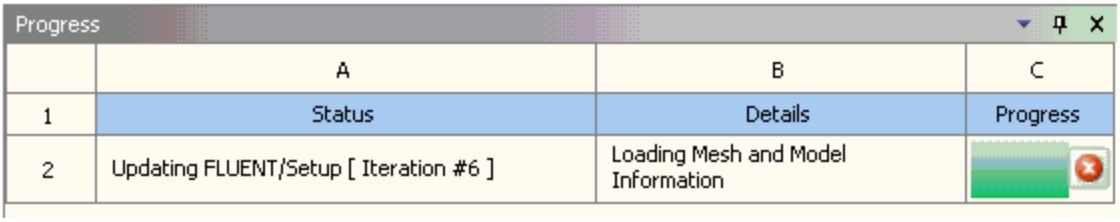
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

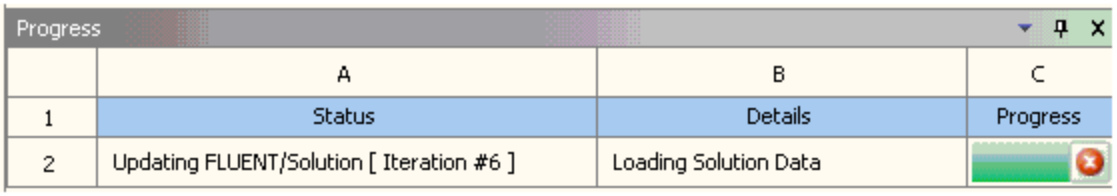
Scenario	Steps to execute	Outcome
components are coupled to the iterator	system	

Starting an Iterative Update

Scenario	Steps to execute	Outcome
Standard update scenario	• RMB on iterator component and select Update GUI operation OR • Update Project OR • Update all design points OR • DX Update	For each iteration, the Electromagnetics product solution is updated followed by the downstream setup and the downstream solution. Each of the client component updates display their progress monitor and can be aborted (resulting in aborting the current iteration)







Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Running a Single Iteration

Scenario	Steps to execute	Outcome
NumCompleted < NumIterations	RMB on iterator component and select Single Iteration Update	One iteration is run. "Completed Iterations" property is incremented.
NumCompleted ≥ NumIterations	same as above	same as above

Interrupting an Iterative Loop

Currently, there is no special progress or interruption control. The progress monitor for individual components displays as they are updated and any control they choose to provide (interrupt, abort or both) is available. Choosing to abort any of the client component updates aborts the current iteration, and the Completed Iterations property remains unchanged from the previous iteration.

Resuming an Interrupted Iterative Loop

Scenario	Steps to execute	Outcome
Single step	RMB on iterator component and select Single Iteration Update	One iteration is always run and the Completed iterations property is incremented.
Run until completion	• RMB on iterator component and select Update OR • Select the Project Update menu option from the toolbar, etc.	If the user specified number of operations are already completed, nothing is done. Otherwise, the remaining iterations are run.

Modifying the Systems Involved in Iteration (Coupled Clients)

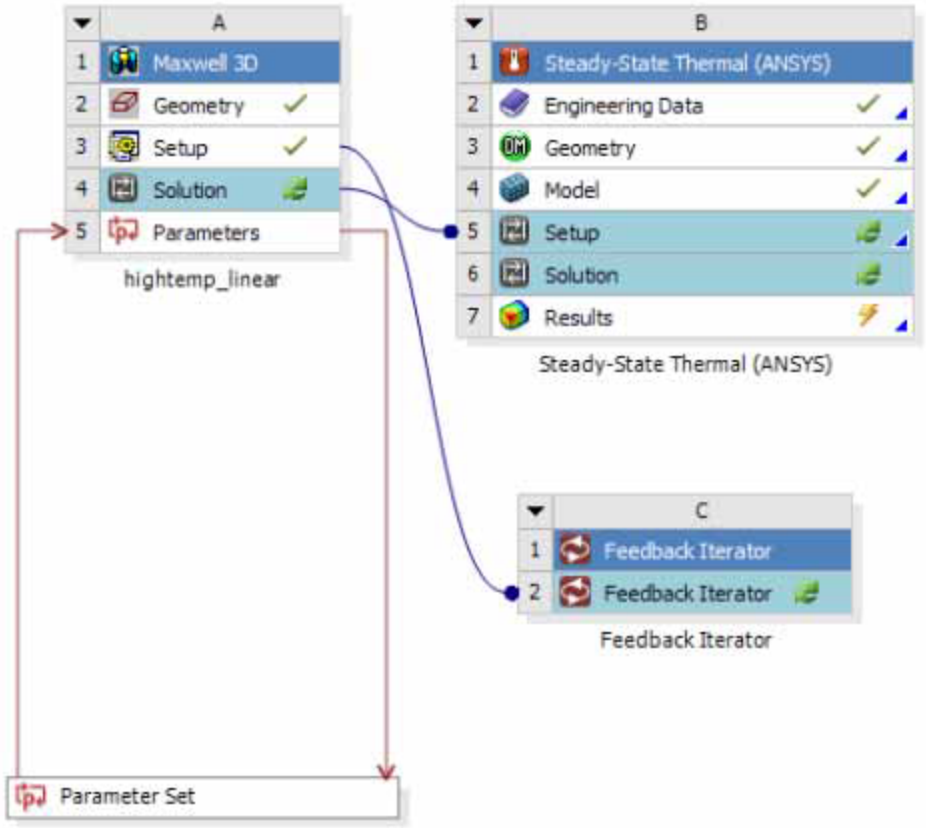
When any coupled client components are modified (either in Workbench or in an external editor), the coupled clients and master are marked as modified. The Completed Iterations property is set to 0.

Iterating to Convergence

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
Standard update scenario	• Right-click on the Feedback Iterator component and select Update OR • Update Project OR • Update all design points OR • DX Update	For each iteration, the Electromagnetics product solution is updated, followed by the downstream setup and the downstream solution. Each of the client component updates display their progress monitor and can be aborted (resulting in aborting the current iteration)

The following temperature convergence example uses Feedback Iterator with a Maxwell design, coupled with a Steady-State Thermal component. Iterating to convergence operates similarly for HFSS projects.

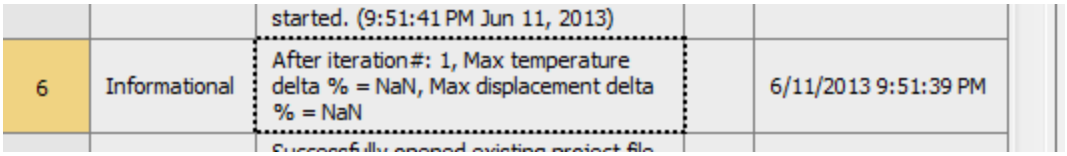


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The example uses defaults of 5% for the convergence targets and a value of 100 max iterations. Recall that the max iterations value is set as a safety measure to ensure that iterations do not continue indefinitely if the solution does not converge. As we progress through the iterations, we observe the following:

End of Iteration #1

The first iteration, because it has no previous iteration, cannot return a meaningful delta value.

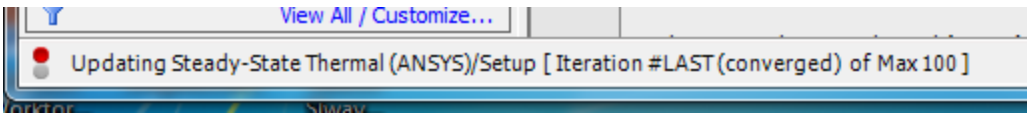


At the end of this iteration, the Latest Delta value is listed as Not Available.

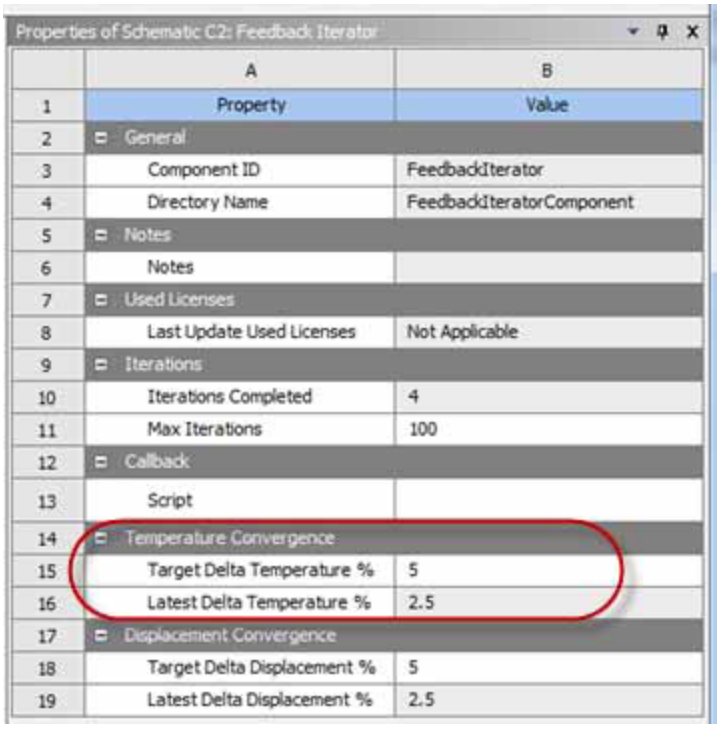
	A	B
1	Property	Value
2	General	
3	Component ID	FeedbackIterator
4	Directory Name	FeedbackIteratorComponent
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Iterations	
10	Iterations Completed	1
11	Max Iterations	100
12	Callback	
13	Script	
14	Temperature Convergence	
15	Target Delta Temperature %	5
16	Latest Delta Temperature %	Not Available
17	Displacement Convergence	
18	Target Delta Displacement %	5
19	Latest Delta Displacement %	Not Available

Convergence Achieved

In this example, convergence has been achieved by the third iteration.

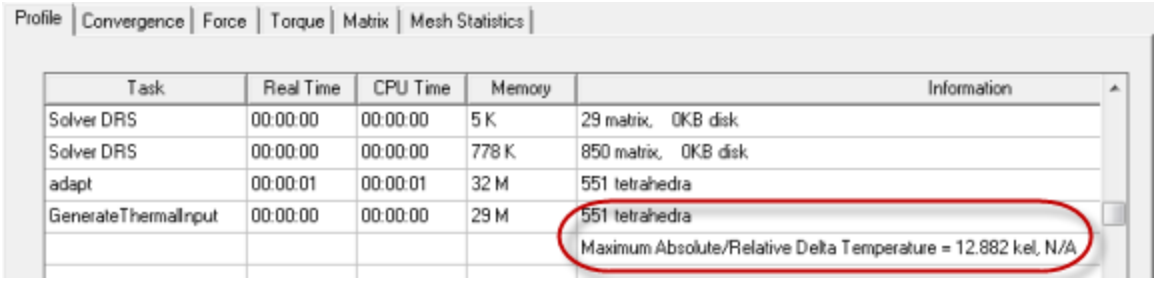


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When convergence criteria are met, the simulation stops. If you attempt to update after convergence has been achieved, the simulation will not be launched since the current solutions satisfy the convergence criteria.

You can monitor the iteration progress by opening the Maxwell editor and selecting **Results** > **Solution data**. Choose the **Profile** tab and keep the window open. As you solve each iteration, you can observe the reported Delta-T (with thermal feedback) and/or delta-displacement (with displacement feedback) and abort the iterations (or stop single iteration updates) when the values reach acceptable convergence levels.



Surface Force Density in HFSS

HFSS can calculate surface force density for Workbench coupling and for post processing within HFSS. The whole approach resembles Maxwell except that HFSS must calculate 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.

force density at the post processing stage while solver provide the surface force density in Maxwell.

The surface force density formulation is provided in a technical note here. Surface forces may exist on surfaces where one side is conductor but the other is not, or finite conductivity and layered impedance boundary. The computation of surface force density will be performed at field geometry instance level.

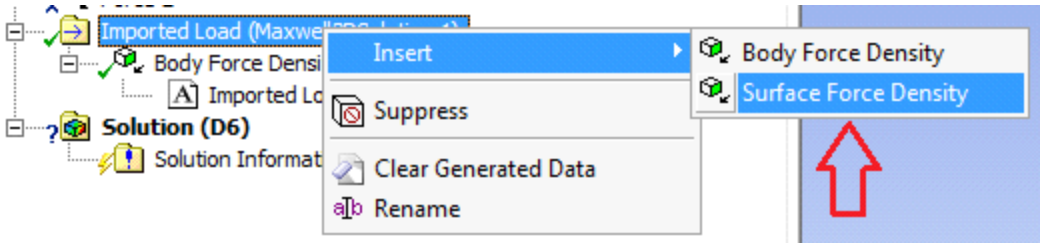
This feature is mainly for the purpose of mapping surface force density in HFSS to Workbench Mechanical. We will also provide it for post processing within HFSS. This feature should work regardless of HFSS solution types as long as surface forces exist. Surface forces may exist when one side is conductor but the other is not, or finite conductivity and layered impedance boundary.

If an old coupling project happens to request surface force and is solved again, then the surface forces (likely nonzero) will be passed to mechanical side.

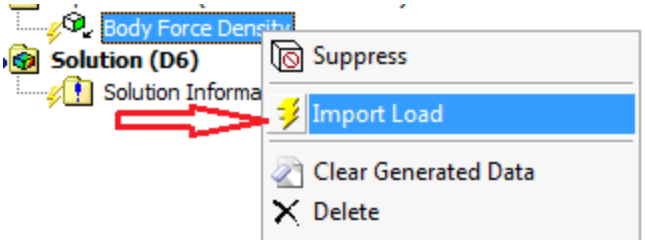
Mapping surface force density from HFSS to Work Bench Mechanical

The behavior is the same as all current couplings.

1. From the Mechanical side set up a surface force coupling to an HFSS design similar to Maxwell.



2. Import the load.



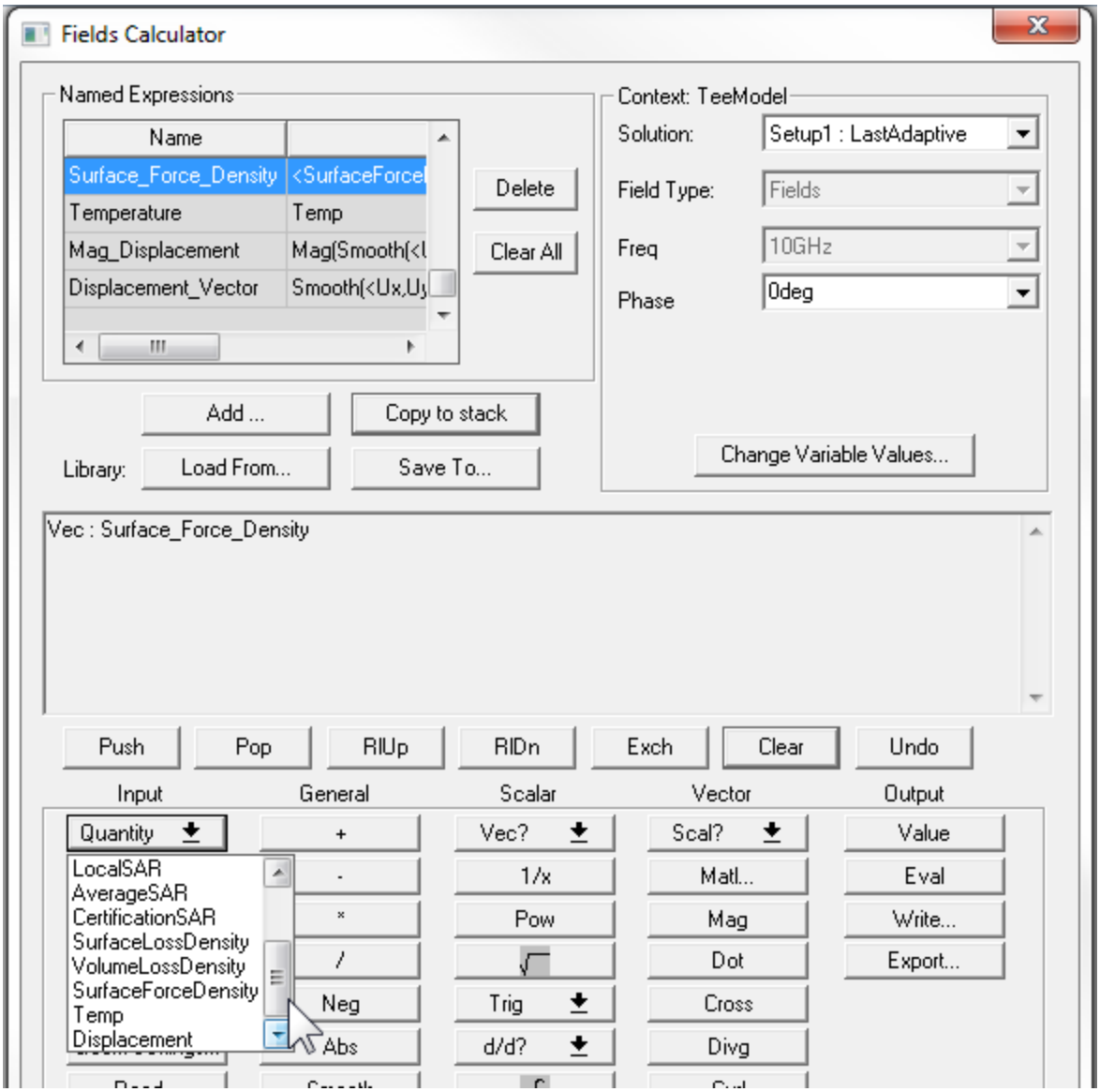
Behind the scenes Mechanical will pass necessary information to HFSS, points on surface to request surface forces and auxiliary files, HFSS will then calculate the surface force densities and pass them back to Mechanical.

3. Post process surface density within HFSS over any surfaces where surface forces exist.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Force Density will appear in the context menu in the Other category under Fields menu.

In the fields calculator Surface Force Density appears in the Named Expression list and the Quantity drop-down menu. So the user can use it the same way as other vector quantities.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17 - Meshing In HFSS 3D Layout

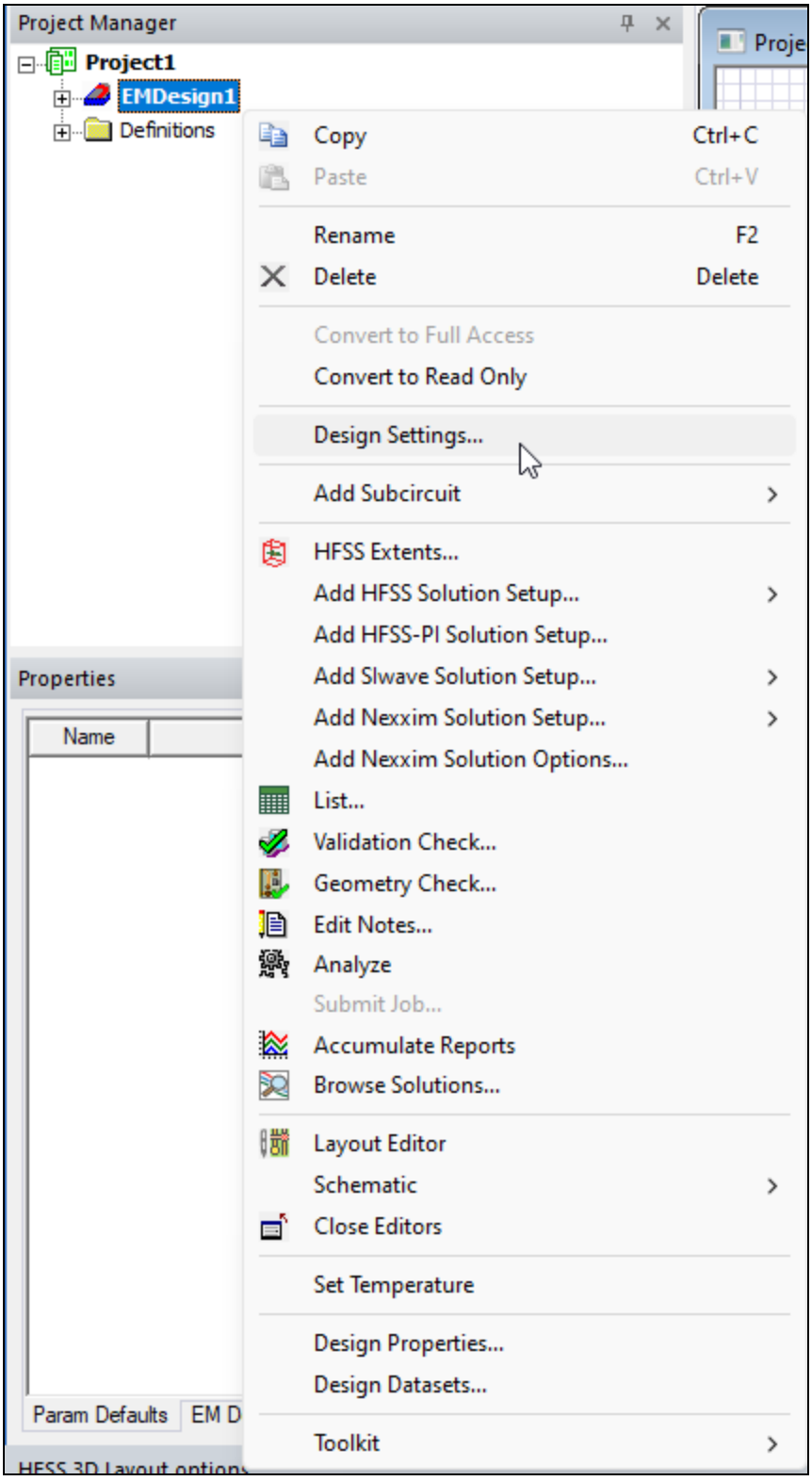
Automatic adaptive meshing is an essential and integral part of the simulation process. An appropriate mesh that supports electromagnetic FEM calculations is essential for obtaining accurate results. Meshes are produced automatically, ensuring efficient simulations in generating highly accurate results. During the simulation process, the initial mesh is created and then the adaptive refinement process begins in which Maxwell's Equations are solved and the mesh is intelligently refined. This adaptive refinement process continues until the solution is converged and accurate.

In IC mode, mesh regions partition a design into multiple domains, potentially increasing mesh robustness and performance for complex layout geometry. For larger designs, enabling mesh regions in an HFSS solution setup allows users to activate parallel refinement, a process that significantly accelerates adaptive passes by refining each region individually to reach a final solution. Defining a mesh region enables users to leverage mesh fusion and parallel adaptive refinement.

To initially access and change meshing methods:

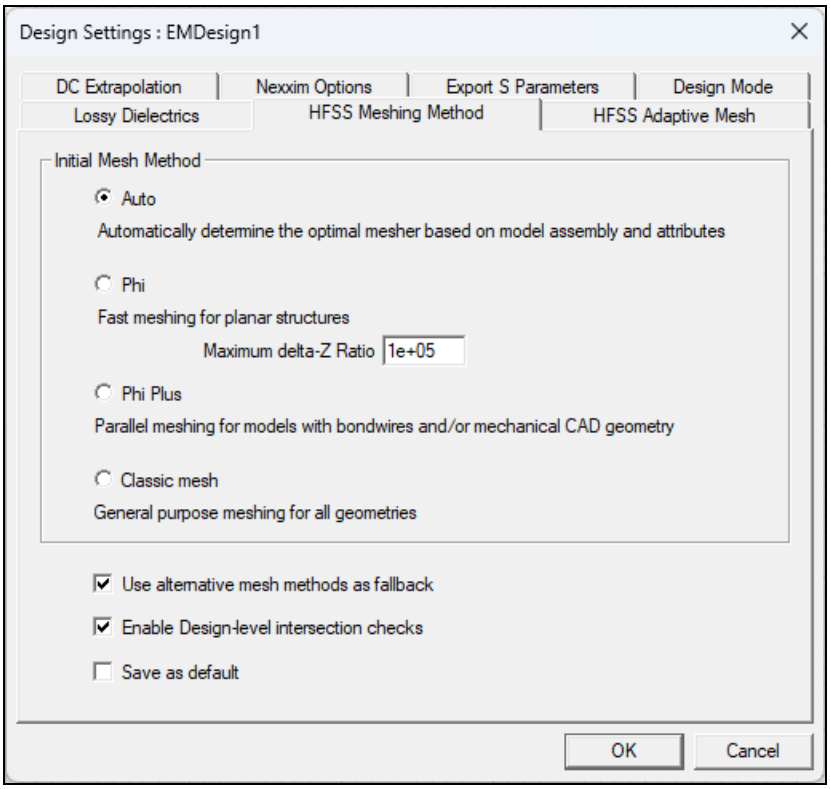
1. From the **Project Manager** window, right-click the active design and select **Design Settings**.

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. Click **HFSS Meshing Method.**



For more information about the concept of meshing and mesh generation, see:

- Introduction to Meshing in HFSS 3D Layout
- Mesh Generation
- Meshing Method
- Initial Meshing Method

For more information about the solution process, see:

- Adaptive Refinement
- Adaptive Mesh Refinement Process Flow
- HFSS Meshing Method Tab

Introduction to Meshing in HFSS 3D Layout

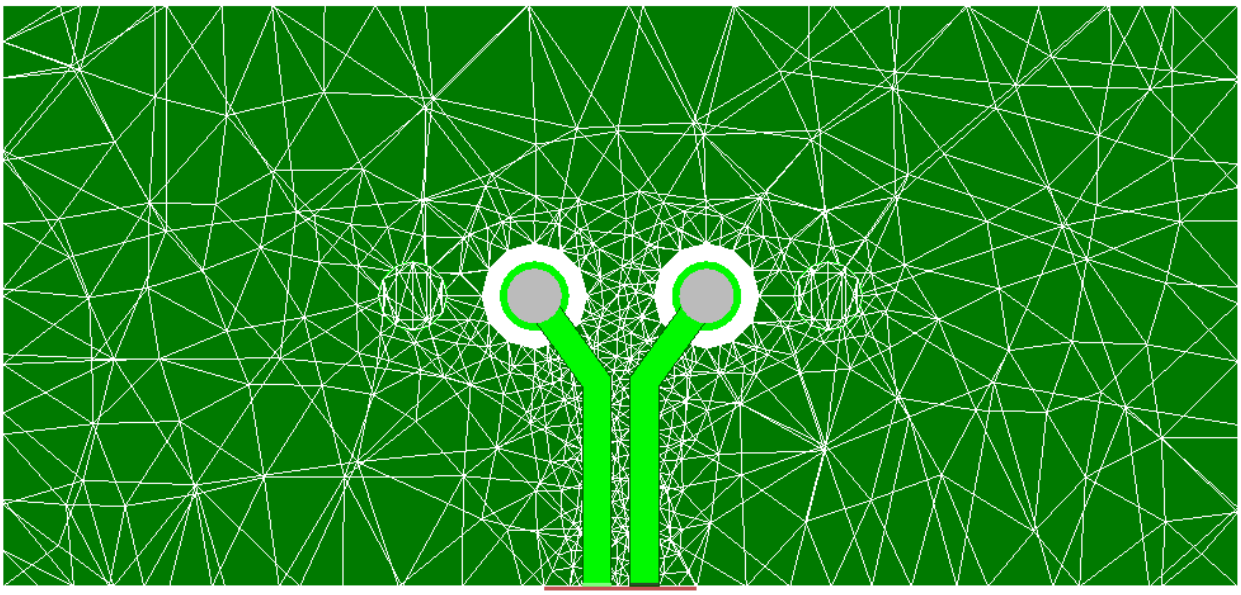
The geometric model is automatically divided into a number of tetrahedra. The collection of tetrahedra forms the finite element mesh.

The partial differential equations that govern the behavior of the electromagnetic fields form an accurate mathematical model. The partial differential equations are Maxwell's Equations, valid

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

everywhere in the three-dimensional space. The finite element formulation uses sophisticated mathematical techniques to satisfy the partial differential equations in the geometrical model, which represents some device or system of interest.

Collection of tetrahedral forming a finite element mesh



Meshing Method

HFSS 3D Layout has four meshers available to generate the initial mesh: the **Auto**, **Phi**, **Phi Plus**, and **Classic** meshers.

The **Auto** selection automatically chooses a suitable mesh method for the design based on geometry characteristics (e.g., if the design contains bondwires, **Phi Plus** is selected).

Phi mesh is an advanced layout-based meshing technology that can produce meshes rapidly and generate fast and accurate results. It is used for planar structures. The **Phi mesher** also includes the **Maximum delta-Z Ratio** field. Typically, if **Phi mesher** is selected, a validation check will analyze all combinations of Z-elevation to determine the delta-Z ratio (i.e., the minimum distance between the model's unique Z-elevations). If the ratio is greater than the maximum delta-Z ratio, the validation check fails (although the model will still proceed through meshing). Entering a different value in the **Maximum delta-Z Ratio** field can allow validation to proceed. Refer to "Meshing With the Maximum Delta-Z Ratio" on page 17-74.

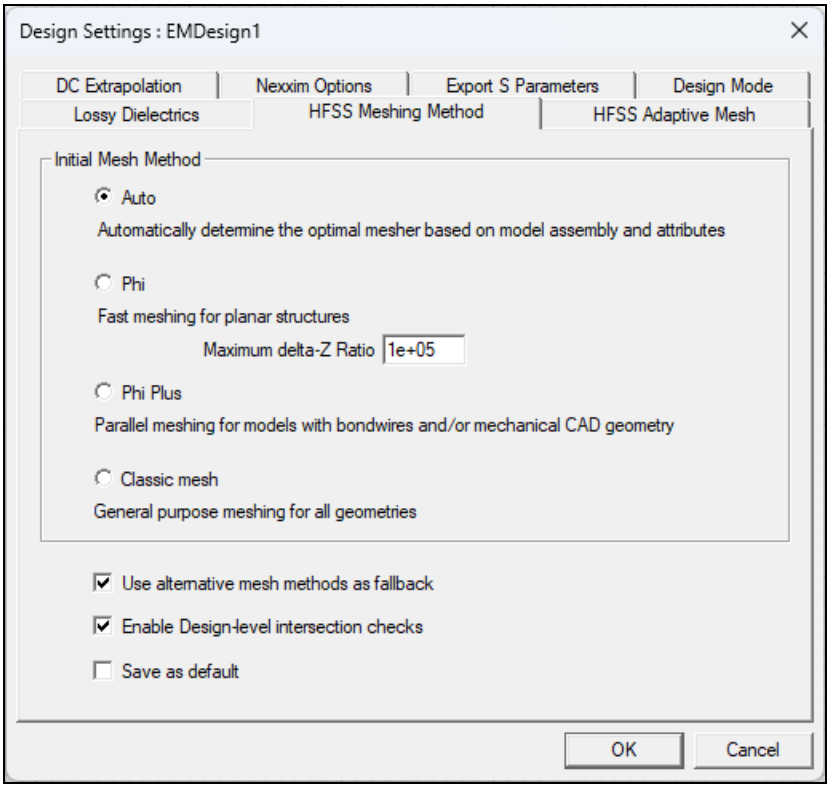
Phi Plus is a multi-domain parallel mesher that works with all types of layout models, particularly those containing bondwires, solder balls, or 3D Components (MCAD).

Classic mesh uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **Phi mesh**, **Phi Plus**, or **Classic mesh** as desired. If **Use alternative mesh methods as fallback** is checked, HFSS 3D Layout uses the selected mesher, if possible, but attempts to use the other meshers if they are more appropriate, based on analysis of the geometry. Depending on the model, **Classic mesh** may converge using fewer resources whereas **Phi** or **Phi Plus** may converge faster. Ansys recommends selecting **Auto** since HFSS 3D Layout automatically determines the correct choice of the mesher for a given geometry.

> **Note:** If **Use alternative mesh methods as fallback** is checked, Phi mesh may not run a validation check on designs that contain certain features (e.g., MCAD components, 3D structures, designs with etching enabled).



Meshing With Rigid-Flex Bends

Adding rigid-flex bends to circuit board designs allows users to reshape the design geometry to simulate realistic mechanical behavior and electrical connectivity. When a design is meshed in a bent state, user-selected meshing methods are overridden (i.e., selections made from the **Design Settings** window > **HFSS Meshing Method** tab). This ensures compatibility between meshing and the deformation imposed by rigid-flex bends.

Meshing selection with active rigid-flex bends is determined 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.

- If the design is Phi-suitable in flat (i.e., unbent) state, it will always be meshed using the Omega mesher when the rigid-flex bend is active (i.e., bent) state. The Omega mesher is available only when the designis completely planar, contains no 3D components, and in a bent state.
- If the model contains any 3D geometry or bondwires, and for all other selections or inclusions, the Classic mesher is applied automatically.

Initial Meshing Method

HFSS 3D Layout designs have four meshers, and five choices, available to generate the initial mesh (i.e., **Auto**, **Phi**, **Phi Plus**, **Classic**, and **Tau**).

The **Auto** selection automatically chooses a suitable mesh method for the design based on geometry characteristics (e.g., if the design contains bondwires, **Phi Plus** is selected. If the Tau mesher beta feature is active, Tau may be selected if the geometry contains large planar facets or small gaps.).

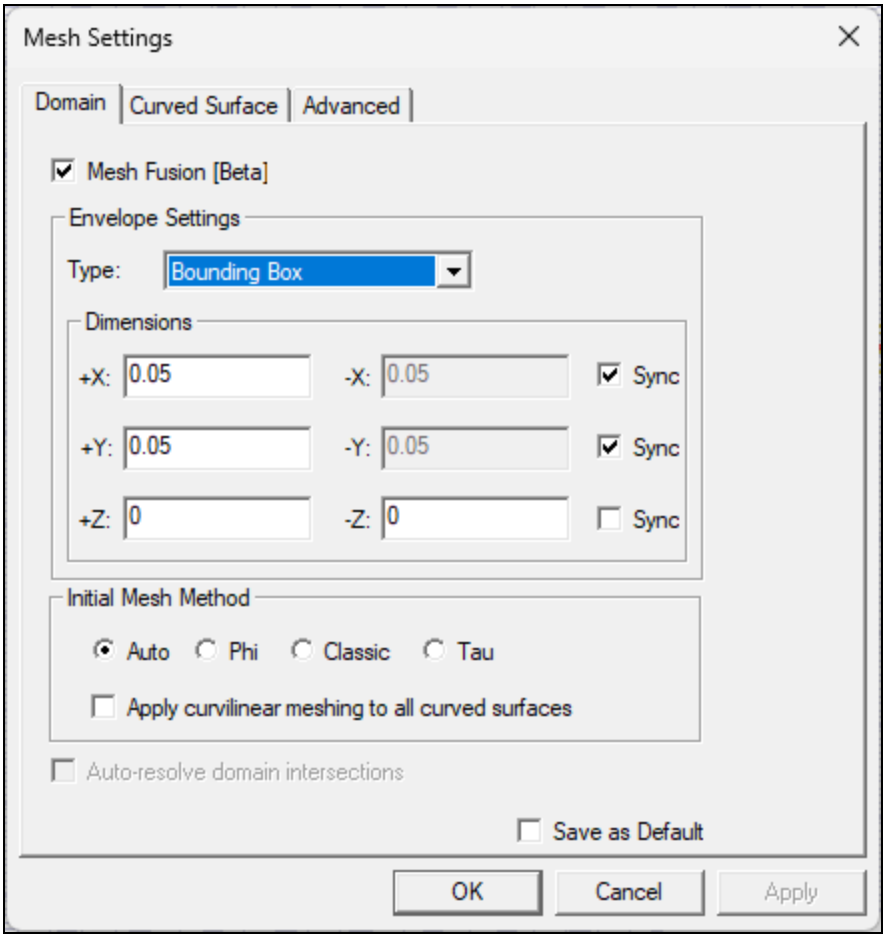
The **Phi** and **Phi Plus** meshers are advanced layout-based meshing technologies that can produce meshes rapidly and generate fast and accurate results. The Phi mesher is a special-purpose fast mesher designed for purely layered structures. In contrast, Phi Plus works with all types of layout models, particularly those containing bondwires, solder balls, or 3D Components (MCAD).

The **Classic** mesher uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately.

The **Tau** mesher is a geometry-based meshing engine designed for curved surfaces and complex structures. It remeshes select curve faces using surface mesh routines and couples them with faceting triangles to ensure a tight, conformal mesh. Tau is preferred for designs with mixed-scale features, large planar facets, or IE Regions requiring high-quality surface mesh. Tau can only be applied to MCAD components. It will not function if the design has ECAD layout geometry.

> **Note:**
>
> The **Tau** mesher is a beta feature specifically for use within a **Mesh Fusion** framework, a hybrid meshing technology that combines tetrahedral and prism-based elements to support mixed-solver workflows. Refer to "Mesh Setup for Mesh Fusion/Assembly" on page 17-26 and/or "Mesh Fusion/Assembly for Hierarchical Design" on page 7-390.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

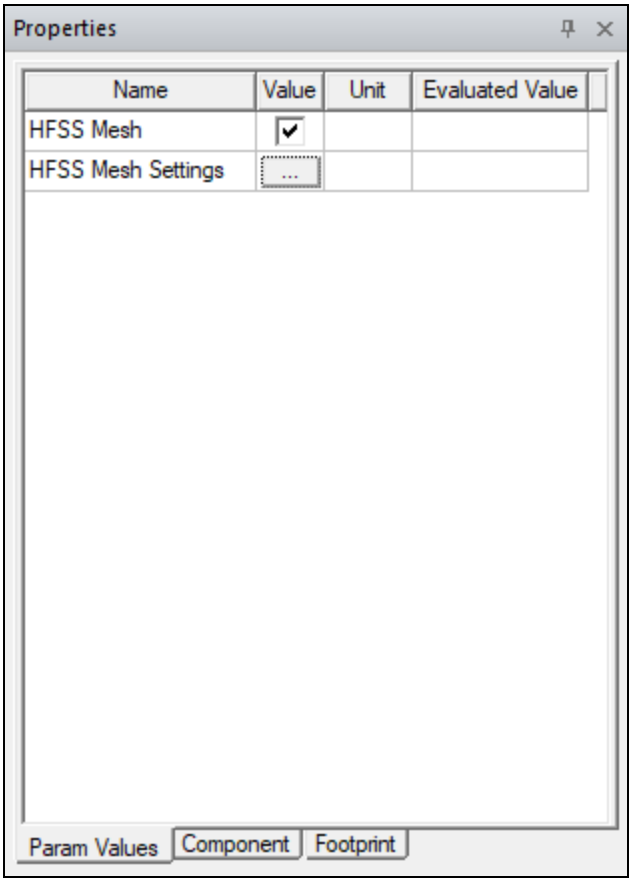


Access the Mesh Settings Window

To choose an initial meshing method, access the **Mesh Settings** window by completing these steps.

1. From the **Layout Editor**, select the active design to make its parameters appear in the **Properties** window.
2. If applicable, click on the **Param Values** tab in the **Properties** window. Then click the **HFSS Mesher Settings** button to open the **Mesh Settings** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

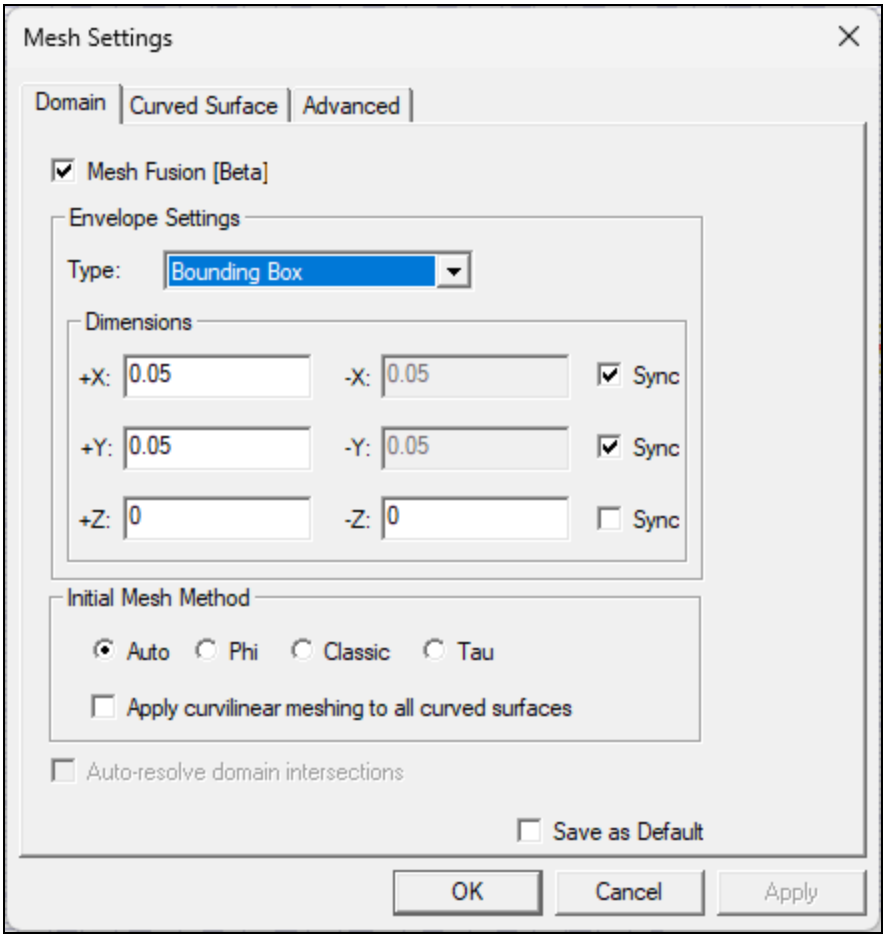


3. Choose one of the following meshers available in the **Initial Mesh Method** area:

   - **Auto** — automatically selects a suitable mesh method for the design based on geometry characteristics (e.g., if the design contains bondwires, **Phi Plus** is selected).
   - **Phi** — a layout-based meshing technology, available in the HFSS 3D Layout interface. This advanced meshing technology is capable of rapidly generating an initial mesh ensuring faster simulations that can be further accelerated and enhanced by using high performance computing.
   - **Phi Plus** — works with all types of layout models, particularly those containing bondwires, solder balls, or 3D Components (MCAD). Phi Plus generates an initial mesh using high performance computing (HPC).
   - **Classic** — uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Tau** — a geometry-based meshing engine designed mesh fusion assemblies.



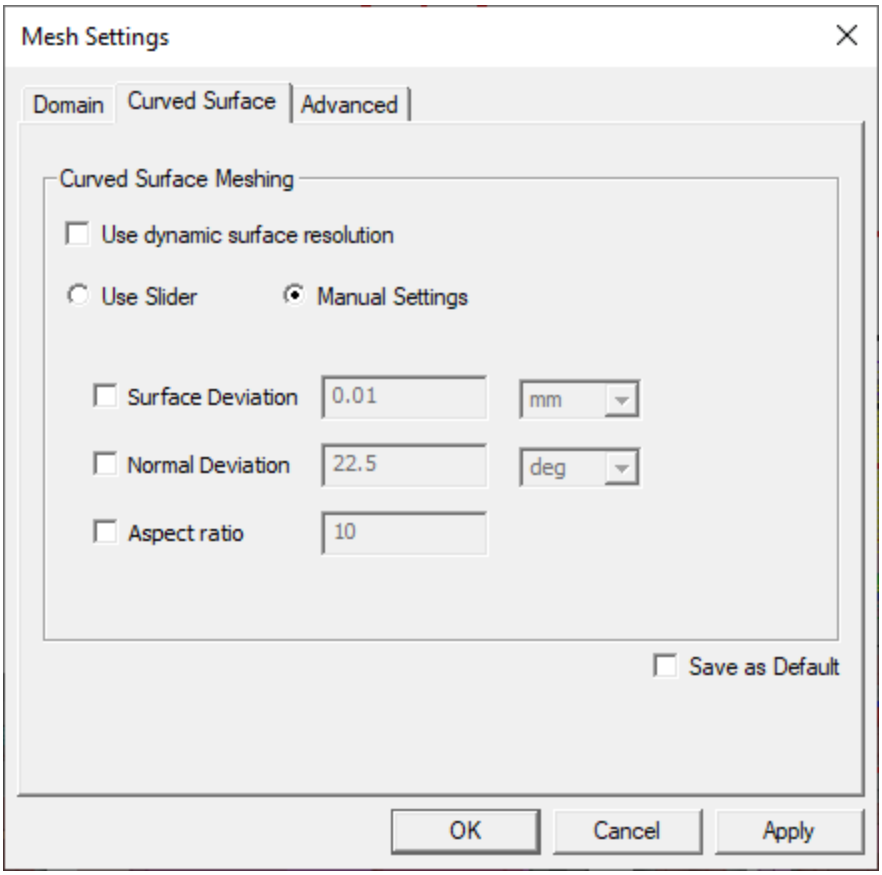
4. Click **Apply**.
5. Click **OK** to close the **Mesh Settings** window.

The procedure is complete.

Defining a Curved Surface Mesh Operation

A curved surface mesh operation is applied to all objects in the model. To illustrate, a model that has two cylindrical objects is used. For curved surface mesh operation, the two important controls for the Manual settings are **Normal Deviation** and **Surface Deviation**. Curved surface operations with respect to these **Manual Setting** controls are described below.

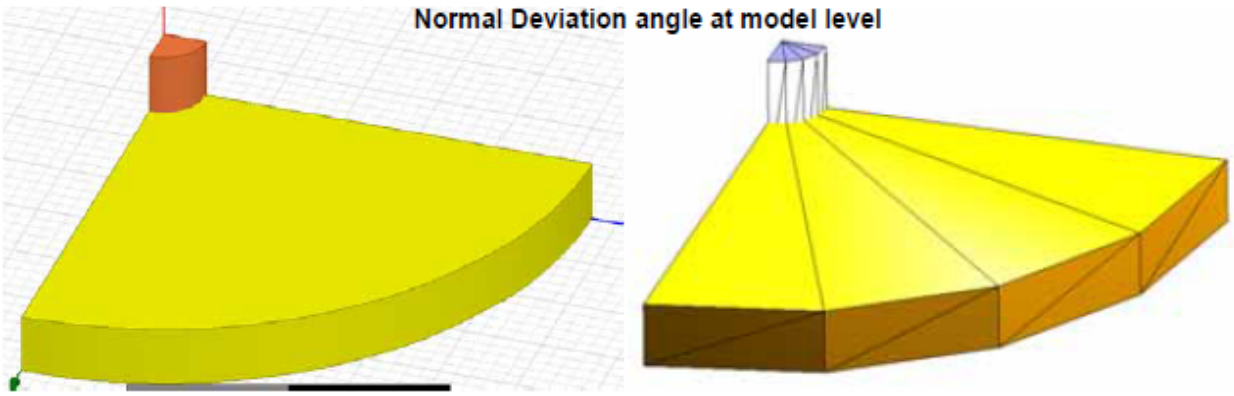
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Normal Deviation Angle

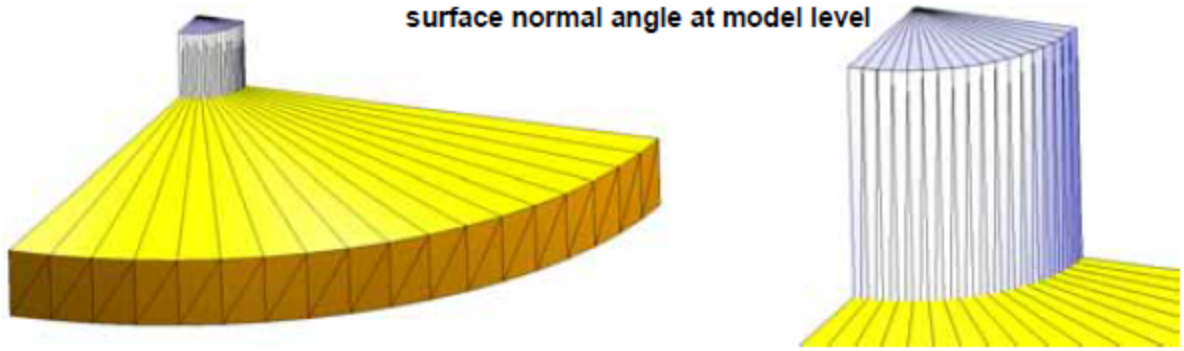
By default, **Normal Deviation** is 22.5 degrees in HFSS. When specifying curved surface mesh operation, consider the sizes and shapes of all curved objects to obtain the desired faceted triangulation model for the mesh.

The following illustrations show a model that has two cylindrical objects.



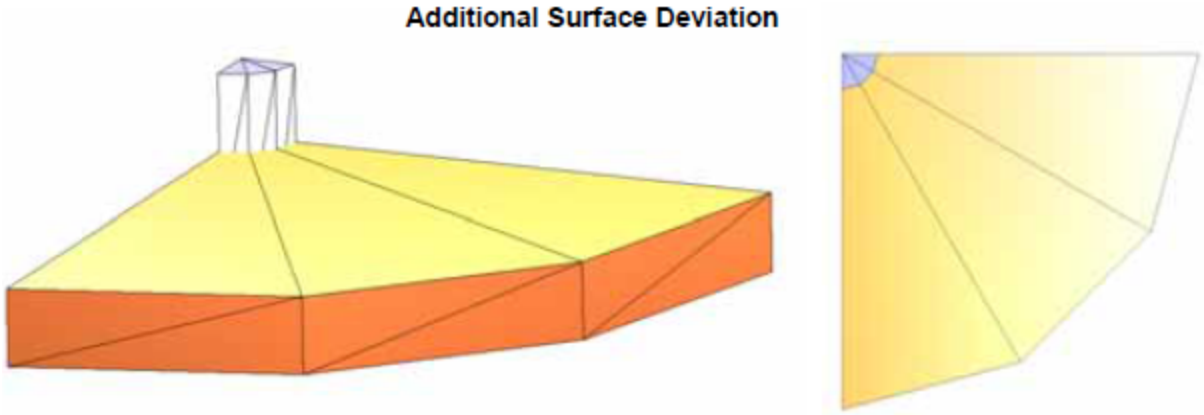
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In HFSS 3D Layout, the 90 degree sweeping curves are faceted as 4 segments since **Normal Deviation** is 22.5 degrees by default. If you specify the surface normal angle as 5 degrees at the model level, there are 18 segments on both cylindrical curved surfaces as illustrated in the following figures. Such a level of granularity is useful for large curved surfaces, but for small curved surfaces, it creates an extremely dense mesh. Therefore, use caution to assign a curved surface mesh operation with small **Normal Deviation** unless all the curved objects in the model have equivalent curvatures and sizes.



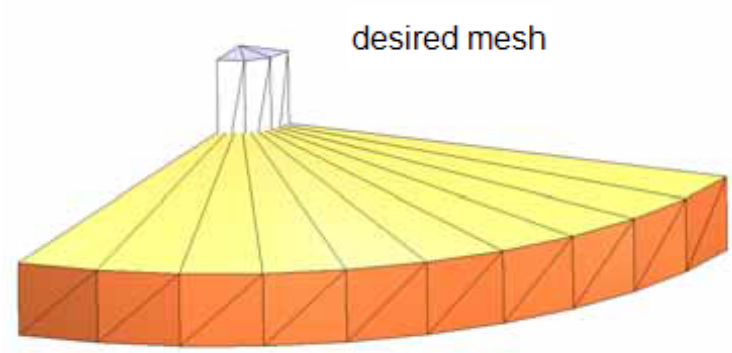
Maximum Surface Deviation Length

An appropriate value of **Surface Deviation** supplements the **Normal Deviation** setting. When you set **Surface Deviation** in addition to the **Normal Deviation** setting, it can further improve the geometry representation. By specifying **Surface Deviation** you can control the difference between the faceted face and the real curved face. **Surface Deviation** is defined in user unit. Consider the same model with two cylindrical objects. In this mode, if you specify 30 degrees as the **Normal Deviation**, there are 3 segments on both cylindrical surfaces as expected.



Since the radius of the large cylindrical surface is 6mm, set 0.06mm as the **Surface Deviation** in the curved surface mesh operation. This ensures we have the desired faceting geometric model.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To summarize, the **Surface Normal Angle** of 30 degrees provides one facetation of the geometry, and the **Surface Deviation** of 0.06 mm controls the difference from the real curvatures. Since the small faceted cylindrical surface already has the difference smaller than 0.06 mm, the surface is maintained as is. However, since the large cylindrical surface has a gap greater than 0.06mm, the facetation is made finer to adhere to the specified length.

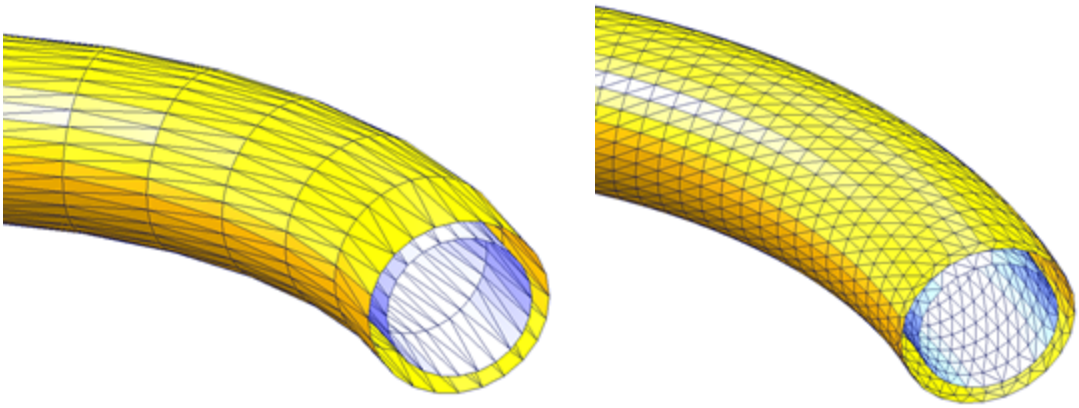
As a rule of thumb, large surface normal angle can control the small curved objects while small Surface Deviation can control the large curved objects in the model-level mesh operation. **Surface Normal Angle** is effective for skewed curved surfaces.

Aspect Ratio

Aspect Ratio *is the ratio of the longest length of an element to its shortest length.* **Aspect ratio** controls the faceting triangle ratio. It is needed only when you want a highly uniform mesh on the curved surfaces or when the curved object is either thin or has a narrow gap. By default, **Classic** mesher uses 20.0 as the **Aspect Ratio** whereas **TAU** mesher uses 8.0 as the **Aspect Ratio**.

When there is a narrow gap between two curved surfaces, a small value of aspect ratio ensures the gap to be uniform. The figures below illustrate a geometric model with the aspect ratio of 20 and 2. Smaller aspect ratio ensures better representation of the curvature for skewed curved surfaces, especially in thin curved layers.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use caution when assigning a small aspect ratio because it will produce a dense mesh at many other curved surfaces that may not require the denseness.

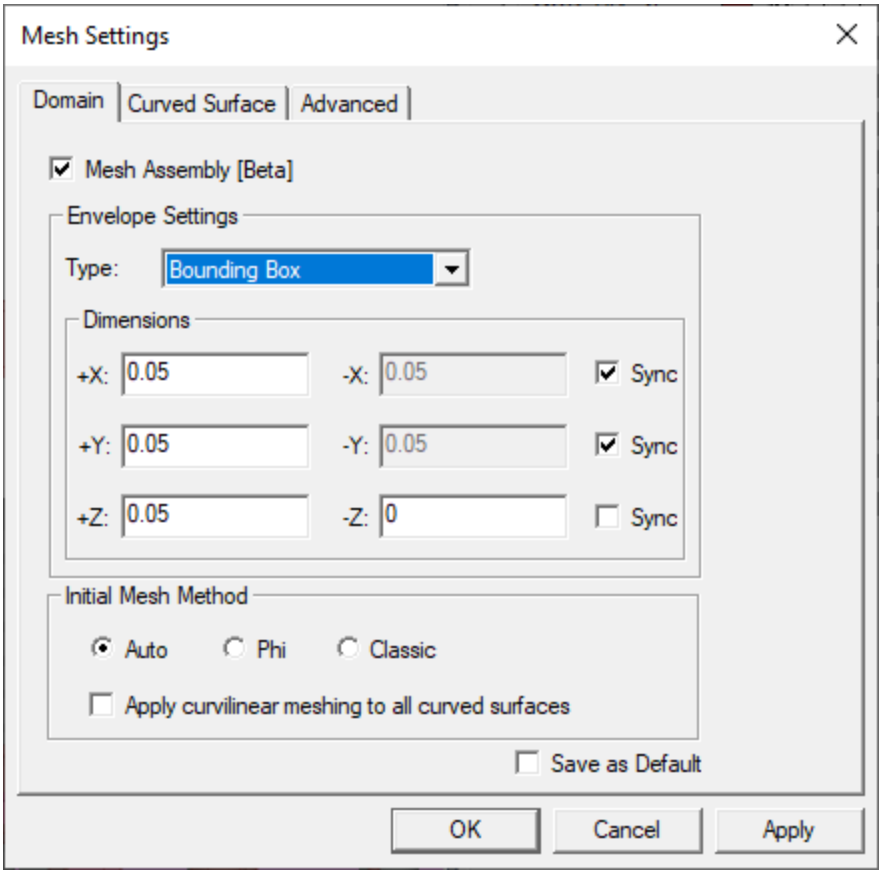
Surface Approximation

Each solver applies surface approximation settings for selected objects when it generates the initial mesh. If you modify a solver's default settings after the initial mesh has been generated, they are affect the mesh for that design variation. Surface approximation makes sense for curved surfaces, for which the mesh are exactly reproduce the surface shape. It also can be used to restrict the aspect ratio of triangles on planar surfaces.

To configure mesh settings for curved surfaces:

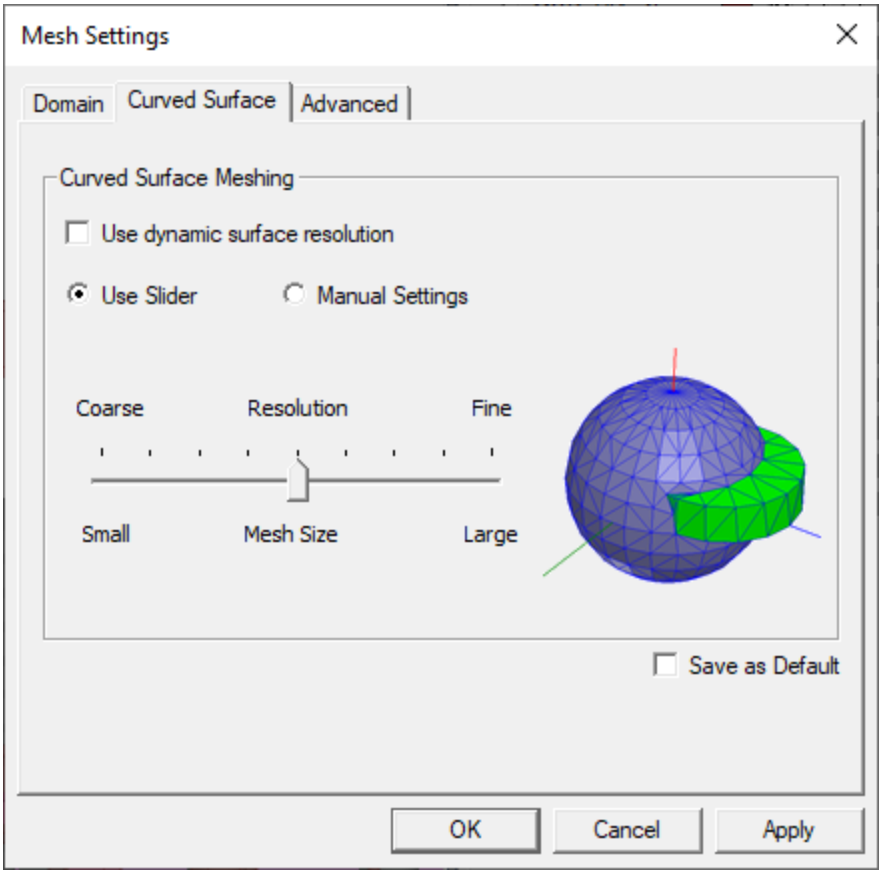
1. Select a 3D object.
2. From the **Param Values** tab of the **Properties** window, click **HFSS Mesh Settings**. The **Mesh Settings** window opens with **Domain**, **Curved Surface**, and **Advanced** tabs.
3. From the **Domain** tab, select **Mesh Assembly** to activate settings in this tab.
4. Select **Apply curvilinear meshing to all curved surfaces** for models with curved surfaces. This increases accuracy, but costs more memory.

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 **Curved Surface** tab:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

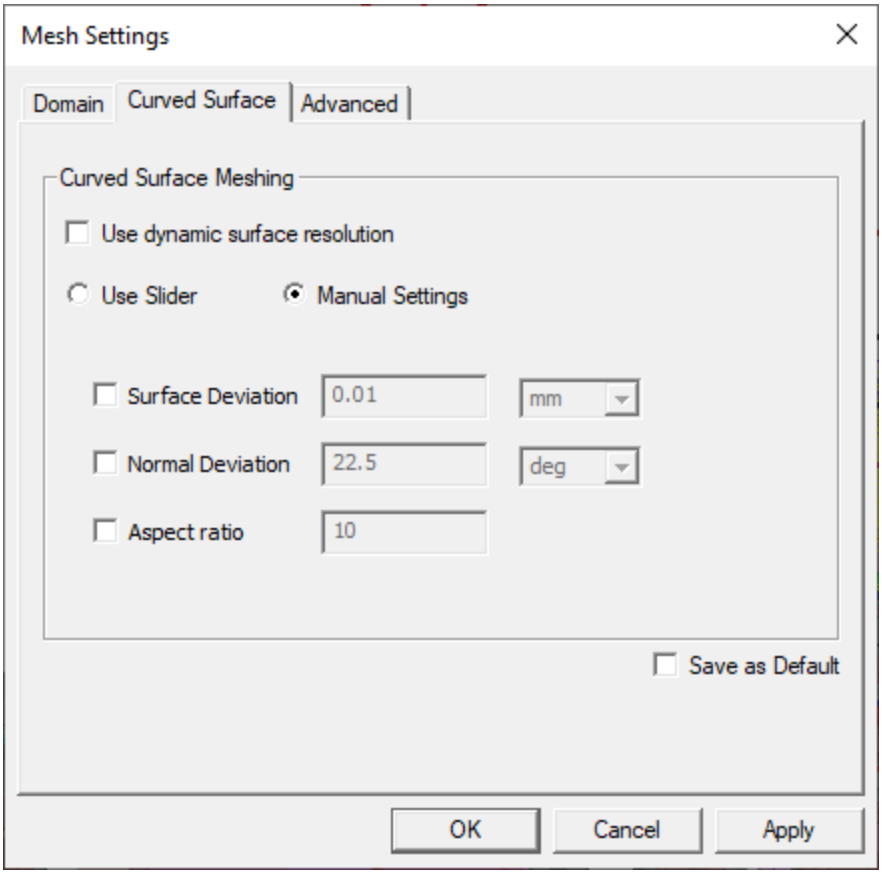


a. Select **Use dynamic surface resolution** to specify the best-practice mesh operations over the geometric models. This mesh operation supports 3D volume mesh and surface mesh in all products. The default mesh operations or user-defined mesh operations may or may not be replaced by the optimized mesh operations with model analysis. In general, large curve faces, curve faces with small gaps, or skewed cables get more smooth curvature representation while small curve faces (e.g., fillets, small curve objects, et cetera), get relatively coarse triangulations so that overall mesh size are reduced. If you select **Use dynamic surface resolution**, the **Manual Settings** option is deactivated.

b. Select **Use Slider** to use the slider with a visual representation of the resolution you select as ranging from Coarse Resolution with a Small Mesh Size through a nine position scale to a Fine Resolution with a Large Mesh Size. The slider bar offers surface representation levels as coarse (1-3), normal (4-6), and fine (7-9). You can also adjust the surface representation level for a specific group of faces by specifying slider bar.

c. If you select **Manual Settings**, the window changes to show fields:

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 check boxes to enable fields and specify values:

- **Surface Deviation** – Use the drop-down menu to select a unit of measure , then enter the distance between the true surfaces of the selected faces and the meshed faces.
- **Normal Deviation** – Use the drop-down menu to select a unit of measure , then enter the angular distance between the normal of the true surface and the corresponding mesh surface.
- **Aspect Ratio** – Enter a value in the field. This value determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 results in well-formed, wide triangles.

> **Note:** When an HFSS 3D component is inside a HFSS 3D Layout design, the aspect ratio must be ≥ 1.

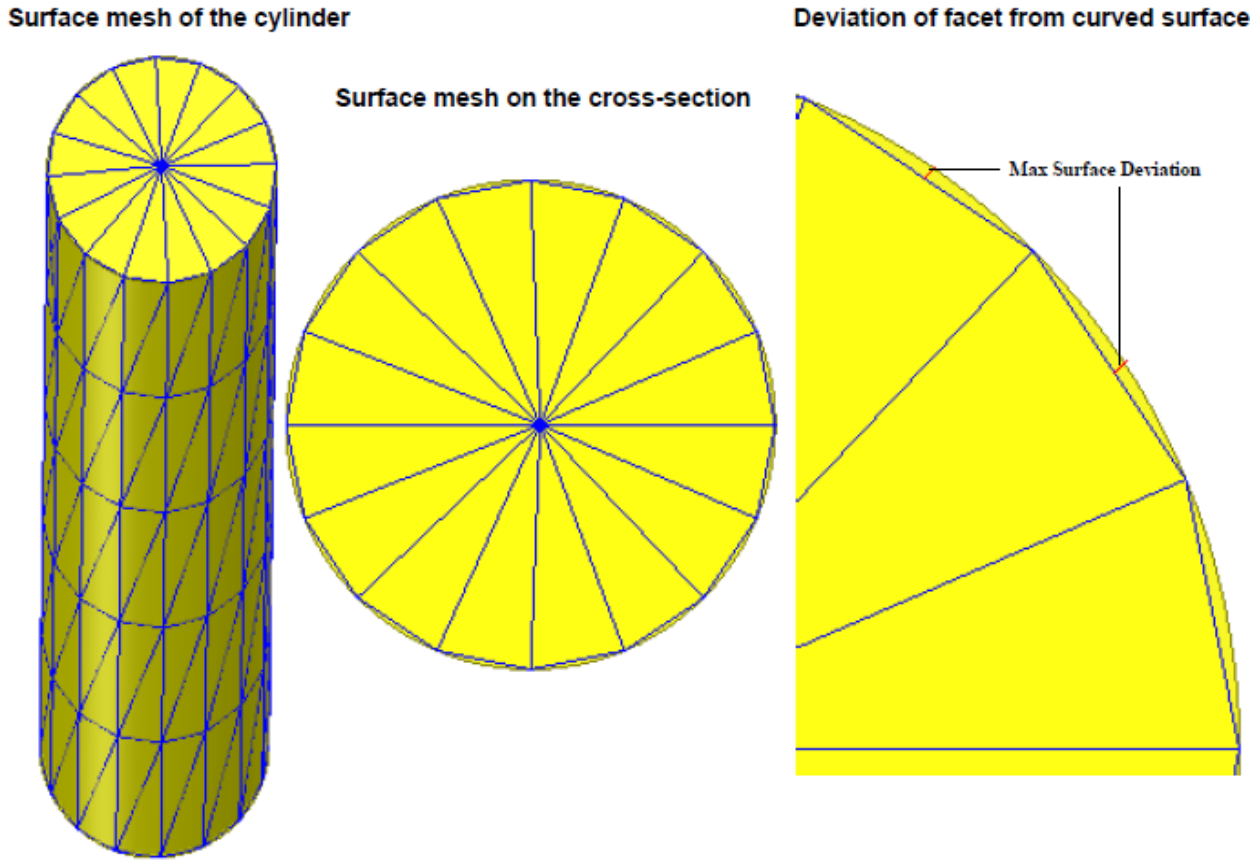
d. To make your choices the default, click the **Save as default** check box.

6. Click **OK** to apply your choices.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

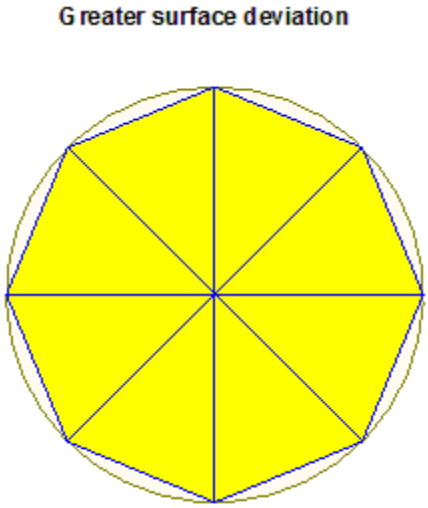
Maximum Surface Deviation (Length)

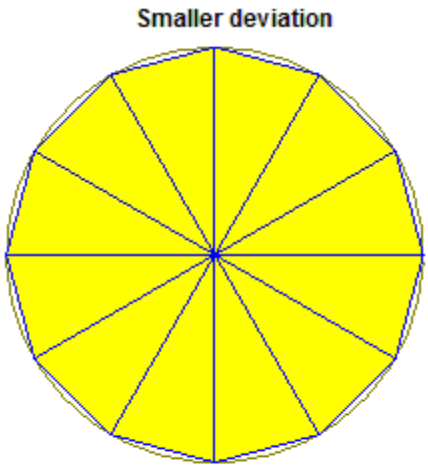
All curved surfaces are faceted during the meshing process. A facet is a collection of triangles. There is a small distance between the faceted surface and the curved surface, which is called surface deviation. **Maximum Surface Deviation** is the maximum value of this distance. The following figures show a meshed cylinder in HFSS, the cylinder cross-section, and the **maximum surface deviation** that occurs near the center of each faceted triangle. When you use an appropriate value for **Maximum Surface Deviation** (length), the curved surfaces are well-represented to realize the original surface.



The following figures illustrate how the representation of a curved surface changes when different values of maximum surface deviation (length) are used. Smaller values of maximum surface deviation can result in greater number of faceted triangulations, for more accurate approximation of the curved surface. However, increasing the number of triangles causes more CPU time and memory consumption during simulation. For this reason, use the default settings, since they are adequate for most circumstances. In the following figure, the cylinder radius is 0.5 mm, height is 1.5 mm, and the normal deviation angle is kept constant at 45 degrees to illustrate the effects of different maximum surface deviation lengths.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

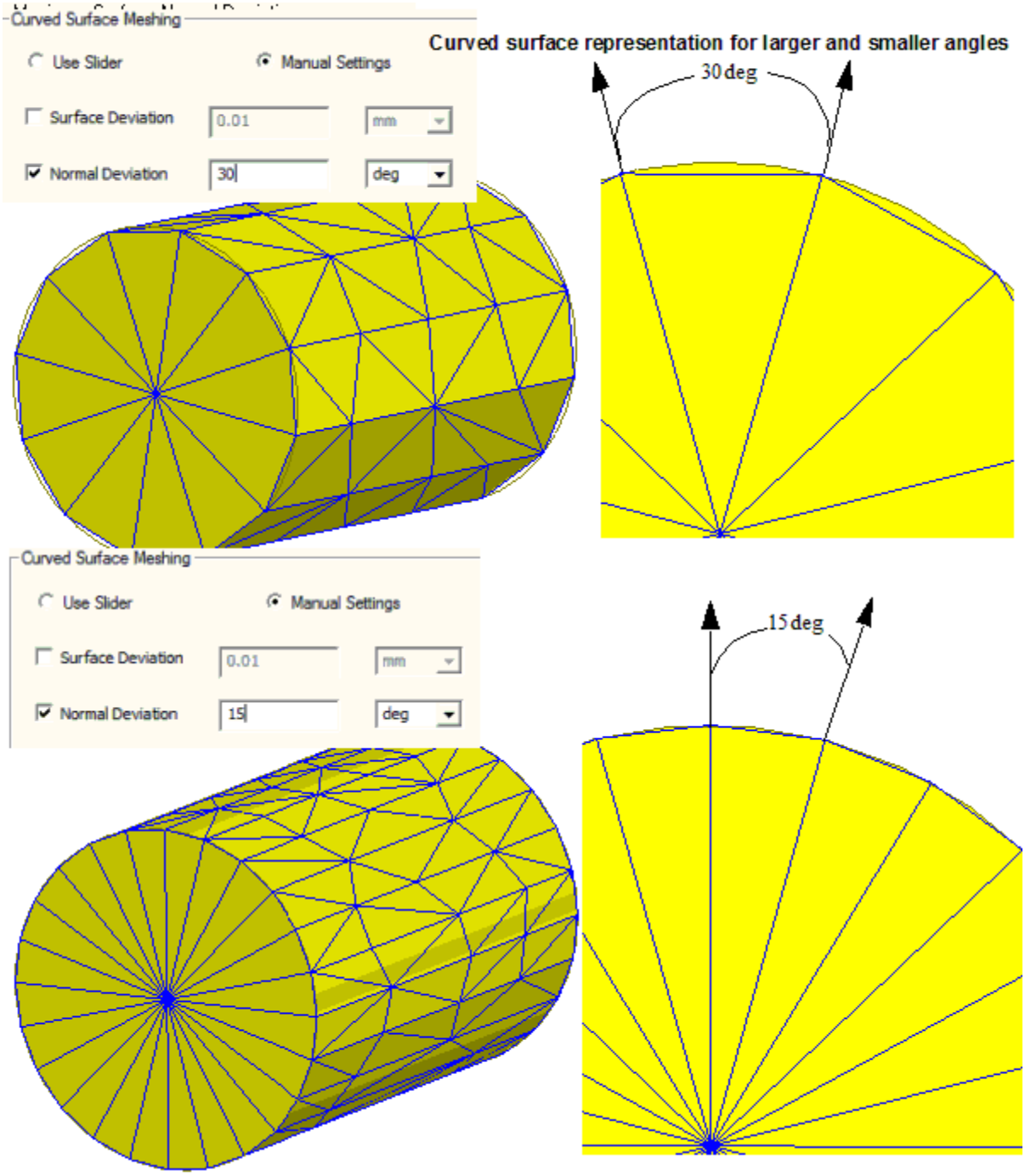




Maximum Normal Deviation (Angle)

Maximum Normal Deviation is the maximum angular deviation between any two normals on a curved surface approximated by a faceted triangulation. The following figures illustrate the effect of using different maximum normal deviation angles on a cylindrical surface. In the following figure, the cylinder radius is 0.5 mm and its height is 1.5 mm.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



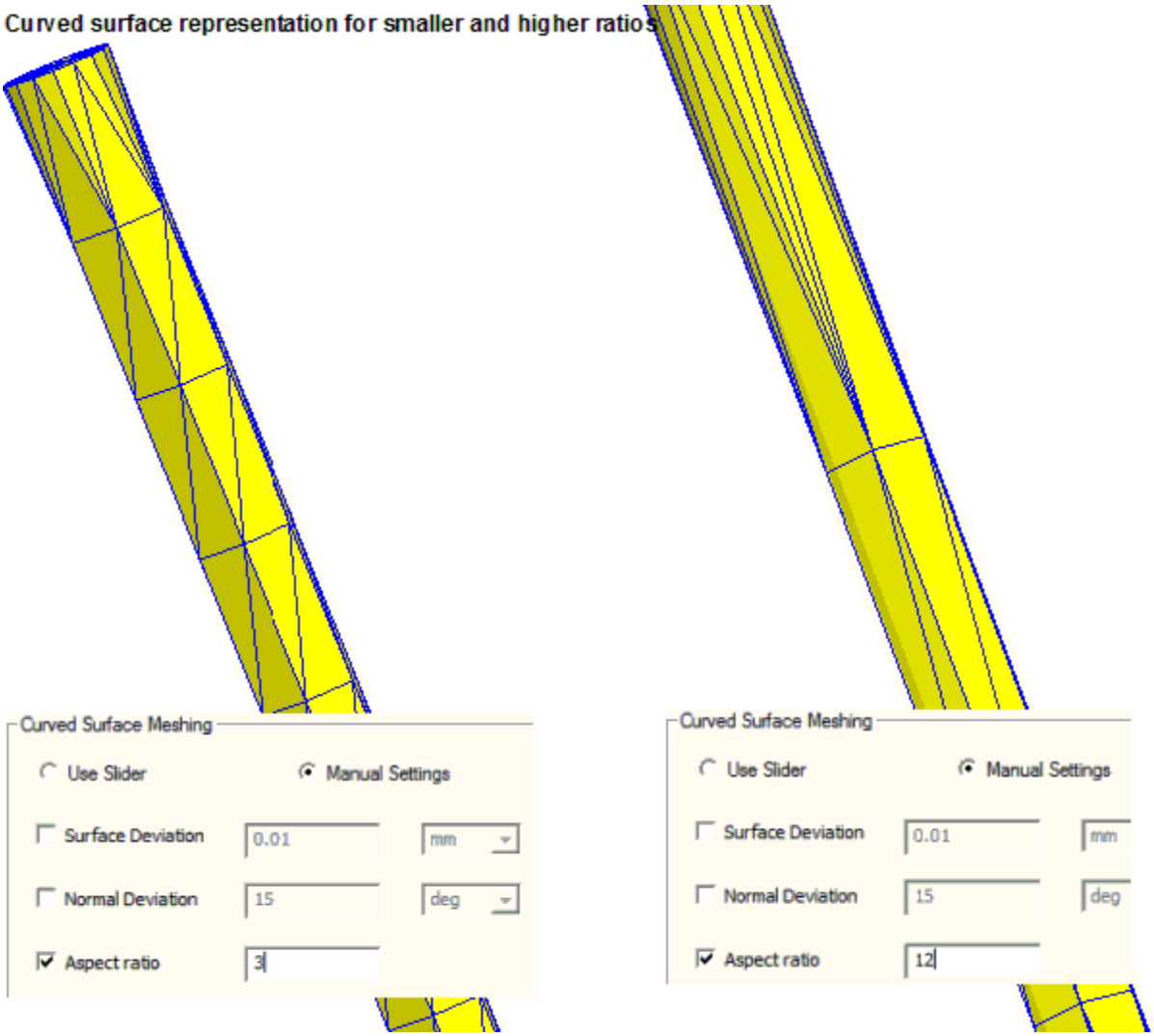
Aspect Ratio

Aspect Ratio controls the ratio of the longest side to shortest side in a triangular mesh. It is a type of mesh operation occurring at the object level, and it directly controls the surface representation of the object. Aspect ratio and the shape of the triangulated facet representing

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 of the object, are related. As aspect ratio increases, the triangles become narrower. Setting aspect ratio to 1 does not guarantee equilateral triangles but produces triangles similar to isosceles right-angled triangles.

> **Note:**
>
> Default value for **Aspect Ratio** setting is recommended. Values in the range of 2 to 20 can produce suitable meshes.
>
> For HFSS and for an HFSS 3D component inside a HFSS 3D Layout design, the aspect ratio must be ≥ 1.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing for Curved Surfaces

For curved surfaces, the initial mesh obtained on the surface of a geometry is the same initial mesh generated in accordance with the surface approximation settings. During the meshing process of curved surfaces, first the volumetric mesh is created based on Parasolid triangulation. For Classic and Phi, the surface triangulations on the volume mesh are very close to the Parasolid triangulation. The approximation can be further improved by modifying surface approximation settings. As the mesh is adapted and refined, new mesh points are added to the surface mesh. If the solution is rectilinear, these points are added to the faceted representation of the curved surfaces. On the other hand, if the solution is curvilinear, then newly added points are used to improve the curved surface representation in the mesh. The process of applying curvilinear elements is often referred to as “pulling points to the true surface,” which essentially means that edges of the triangulated facets are moved closer to the true surface, resulting in accurate representation of the curved surface.

Model Resolution Length

Model Resolution Length sets the threshold for the mesh maker to ignore features smaller than the default or user-specified length and captures major features in the model that are important for the simulation. Models that are imported from other sources may have extraneous details that do not contribute toward the electrical characteristics in the simulation process. If you specify an appropriate value for model resolution length, the mesh maker can selectively remove even tiny unneeded features. When you define the model resolution length for one or more objects, the mesh maker removes the triangulations for these objects.

For objects with uniform thickness or with constant cross-sections, the value specified for **Model Resolution Length** can be less than 40% of the thickness. The default **Model Resolution Length** value is 10e-4 user units. For example, if the model is drawn in mm, all gaps, stubs, holes and other features smaller than 0.0001 mm can be removed by HFSS during the creation of the mesh. It is important that you choose units and values in such a manner that all important and critical features of the model are strictly greater than 0.0001 model units.

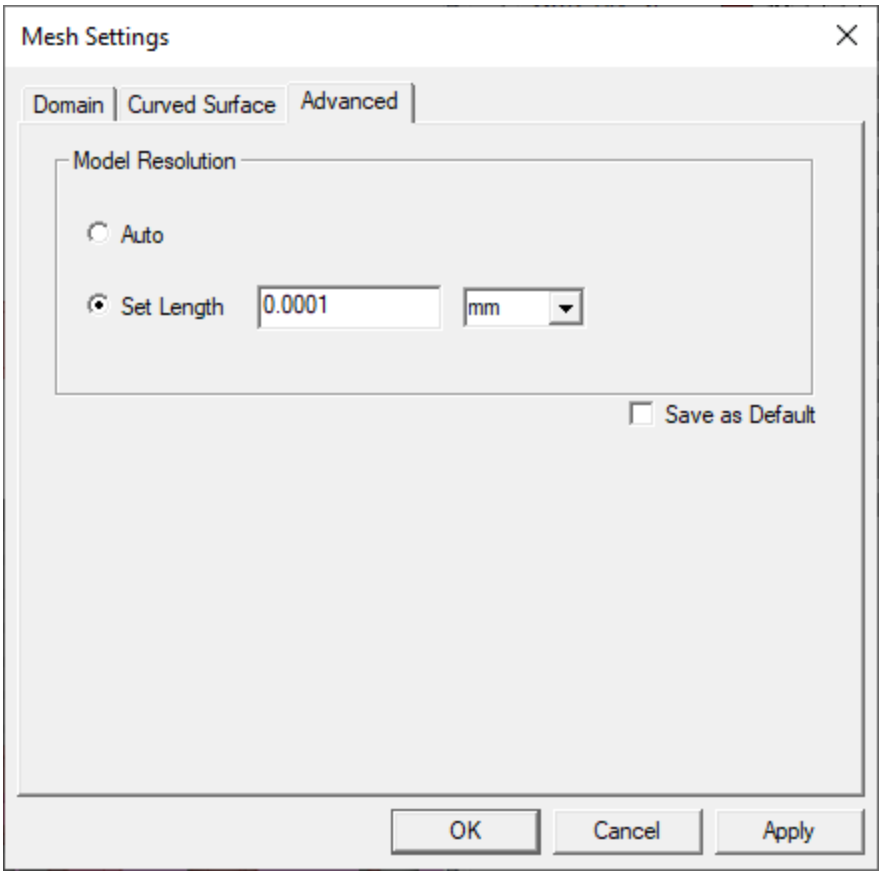
> **Note:**
>
> If you modify the model resolution length operation, the initial mesh is invalidated and HFSS triggers regeneration of the initial mesh.

To set the model resolution length:

1. In the Beta settings, enable **HFSS FEM Mesh Assembly**.
2. Select a 3D object.
3. In the **Param Values** tab of the **Properties** window, click **HFSS Mesh Settings**. The **Mesh Settings** window opens with **Domain**, **Curved Surface**, and **Advanced** 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. On the **Advanced** tab, set the **Model Resolution**. This sets the optimal length of the hypotenuse for triangles in a mesh.

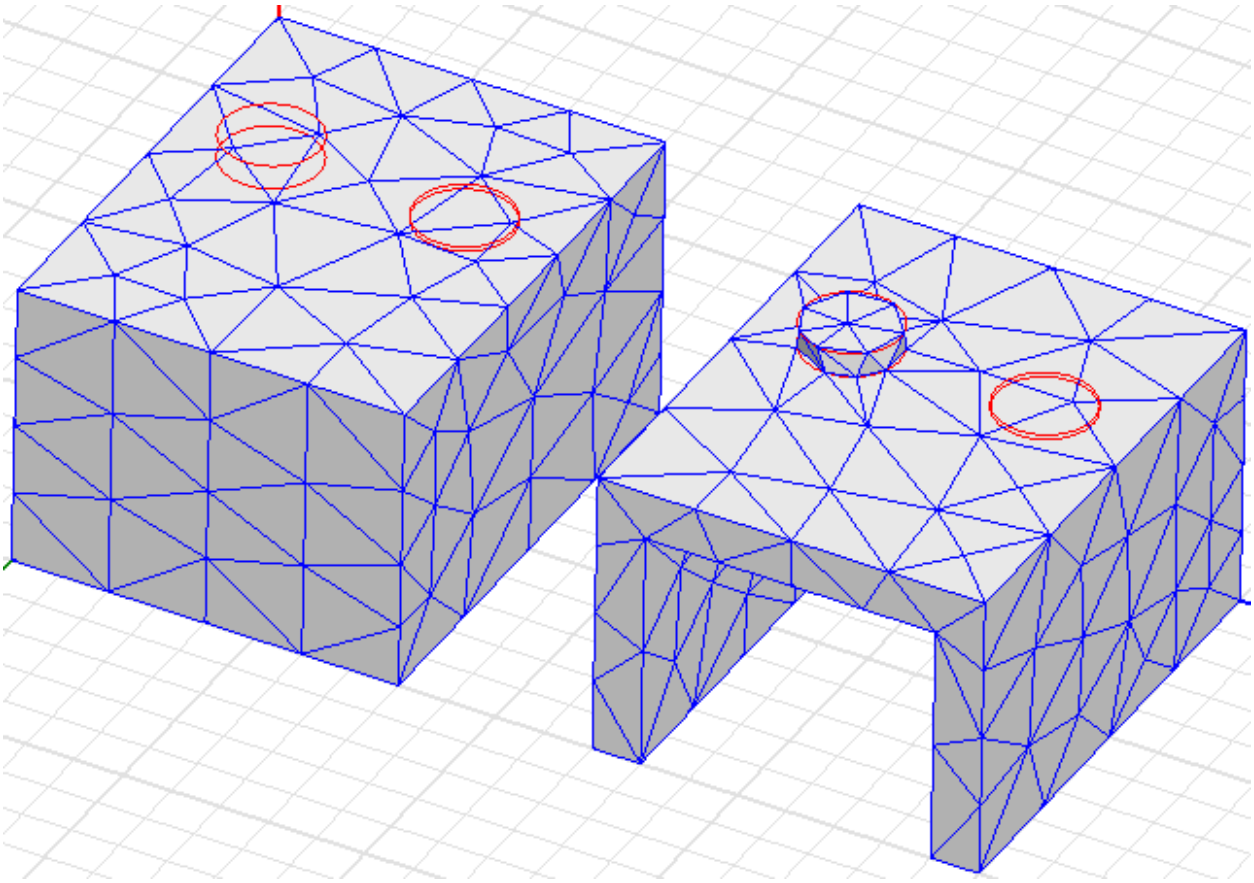


5. Click **Auto** to allow the mesher to determine the optimal length.
6. For a precise length, click **Set Length** and enter a physical distance with units.
7. To save this setting as a default for future models, click **Save as Default**.
8. Click **Apply**.
9. Click **OK**.

Auto Option

The following figures show the results of using the **Auto** option on a model containing a box united with two cylindrical objects. The **Auto** option appears in the **Advanced** tab of the **Mesh Settings** window. The effective thickness of the body in the left is very high. Therefore applying automatic MRL defeatures both the cylinders out. The effective thickness of the body on the right is smaller so it defeatures the smaller stub and leaves the bigger stub.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

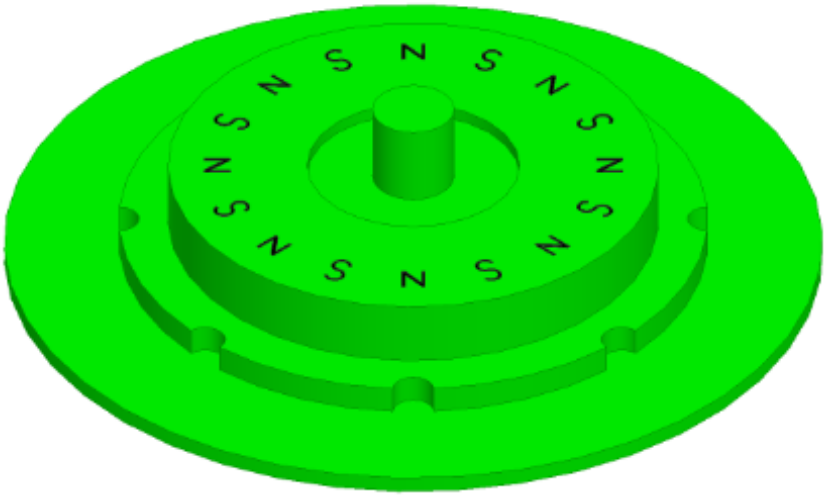


An MRL Scenario

When you define a **Model Resolution LengthOperation,** only major portions of the model are retained in the mesh. An MRL example is shown in the following figure: 3 D symbols S and N are stamped on the top surface of the original geometry.

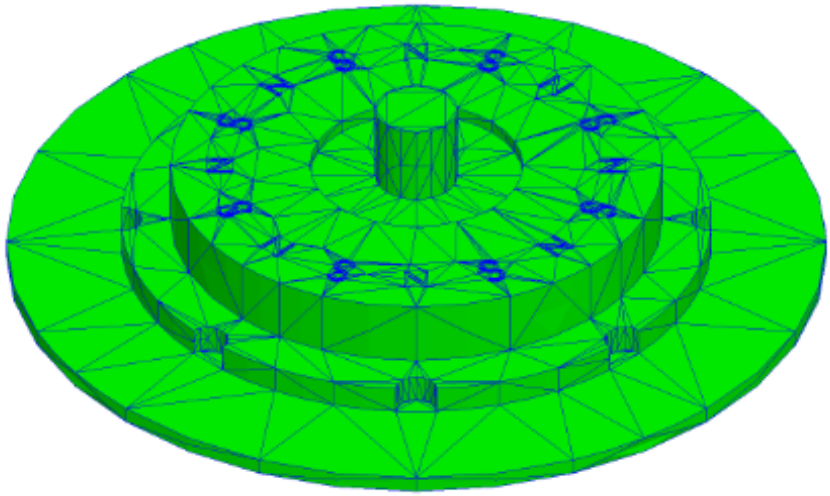
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Original Geometry

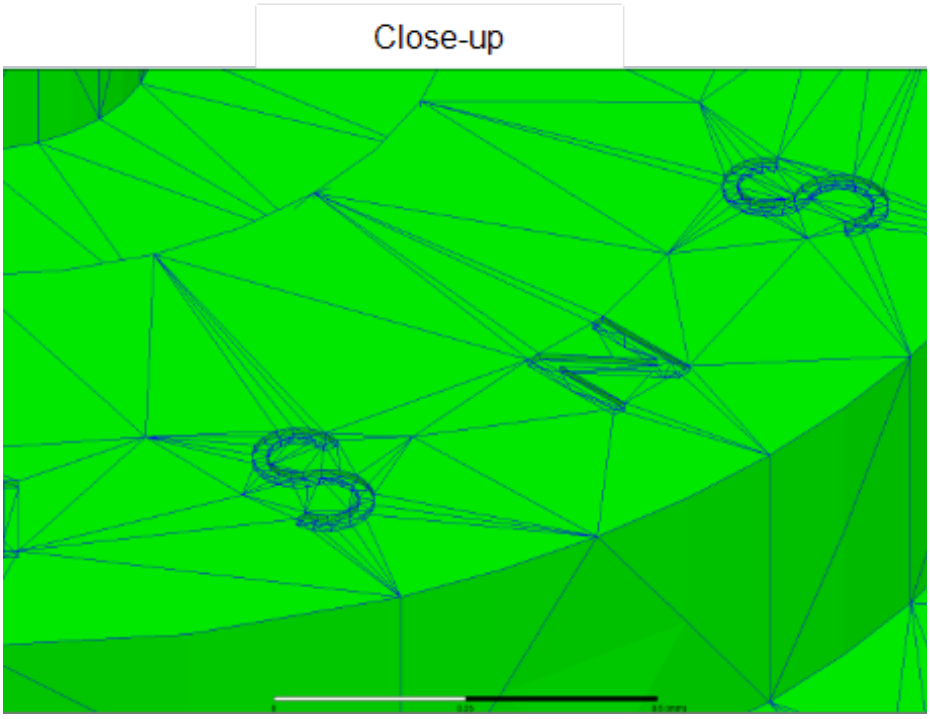


The following figures show the mesh plot without using **Model Resolution Length Operation**. The symbols S and N are magnified to show the mesh's appearance when no **model resolution lengthoperation**is applied. These tiny indentations on the top surface are actually not significant for the simulation. Their presence results in a denser and bigger mesh of the model, especially in the tiny regions as shown below.

Mesh with no MRL



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

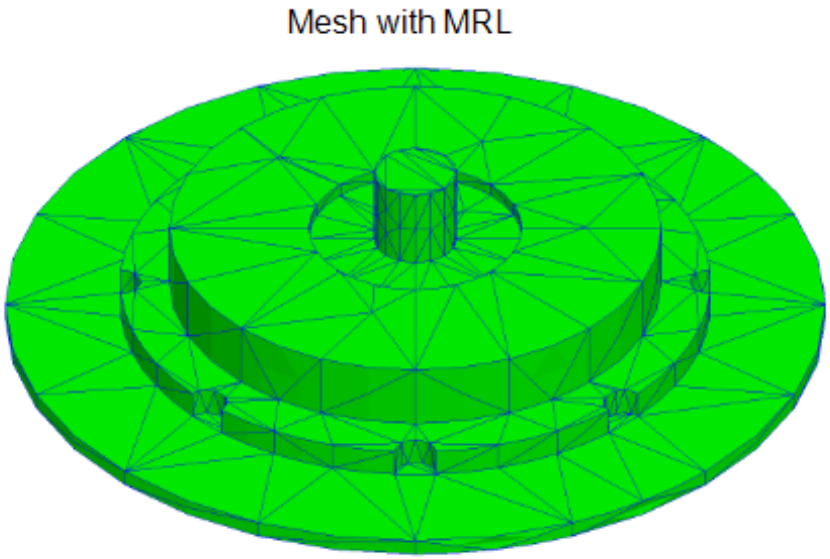


By defining a **model resolution length operation**, these tiny features or indentations can be removed from the mesh.

The following figure shows the mesh of the same model looks when you specify **model resolution length**. The mesh is coarse when compared to the previous mesh. Due to the **model resolution length operation,** the tetrahedrons representing the symbols for north pole and south pole have been removed.

Compare the figures and observe the difference between the meshes before and after the **model resolution length operation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The removal of extraneous features improves mesh quality and simulation efficiency and requires fewer computational resources.

Operations Refining Initial Mesh

Seeding, port, lambda, and adaptive refinement modify the initial mesh. If you modify seed refinement, the changes are applied to the initial mesh saved by HFSS.

Notwithstanding the availability of **Mesh Settings**, use the automatic adaptive refinement technique of solving problems in most cases since it is sufficient to yield accurate results.

Mesh Setup for Mesh Fusion/Assembly

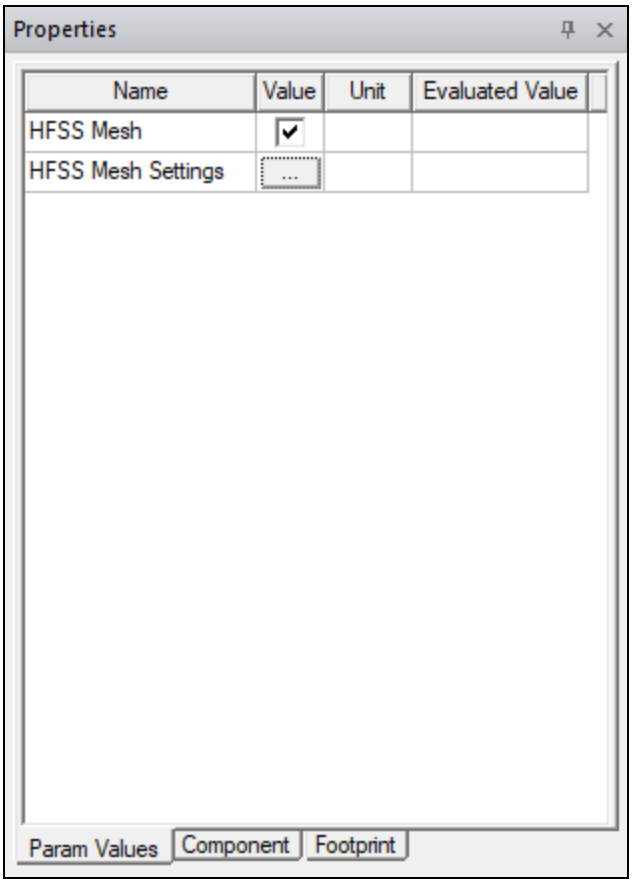
Specify mesh settings, including the surface approximation and the meshing approach. Mesh settings apply to all objects in the given component; however, if separate surface approximations are applied to specific objects within a 3D component, the object settings take precedence over the general settings. For most designs, the solver automatically selects the most selected meshing approach. Otherwise, configure options in the **Mesh Settings** window.

> **Note:**
>
> Mesh Regions can be created in IC mode. Refer to Creating Mesh Regions and Activating Parallel Refinement (IC Mode Only).

Follow these steps to access the **Mesh Settings** 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. From the **Layout Editor**, select a 3D object in the active design to make its parameters appear in the **Properties** window.

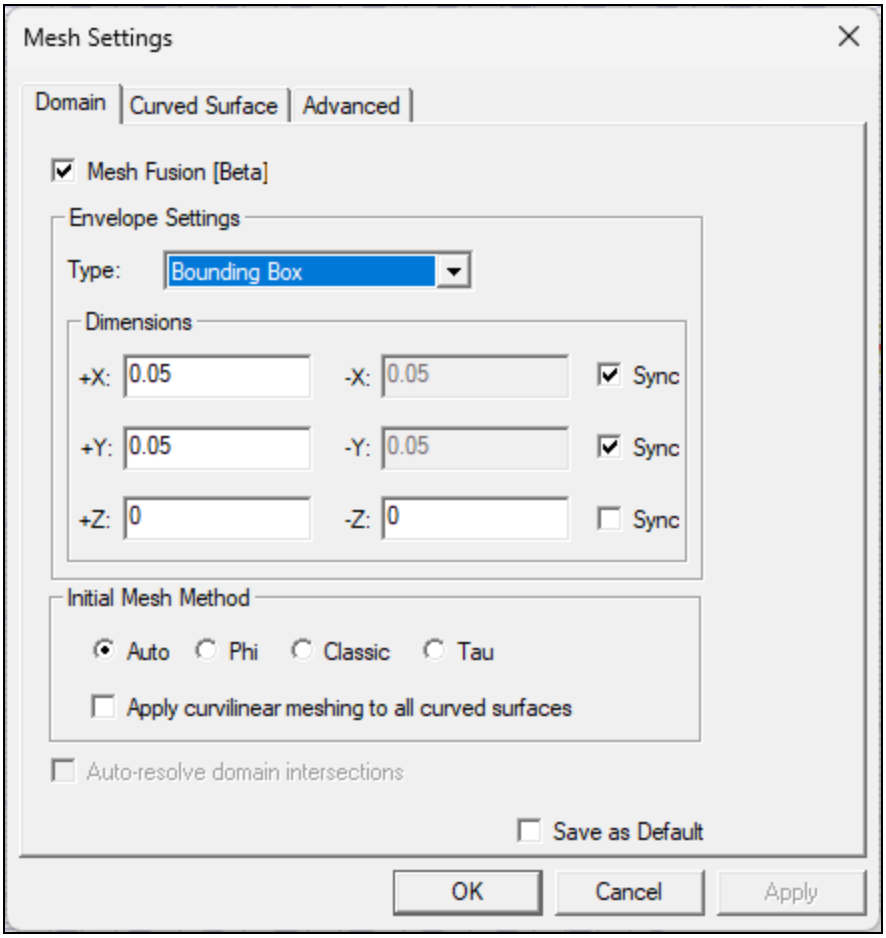


Note:

Mesh Fusion is supported **only** for layout and 3D components and for Component 2.0 representations. If a component does not meet this requirement for mesh fusion, the option is not available on the **Param Values** tab.

2. Click the **HFSS Mesher Settings [...]** button to open the **Mesh Settings** window with **Domain**, **Curved Surface**, and **Advanced** tabs.

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 necessary, check the **Mesh Fusion [Beta]** box to access the **Envelope Settings** and **Initial Mesh Method** group boxes.
4. From the **Envelope Settings** area:
    a. Select a **Type** of bounding from the drop-down menu:
        - **Bounding Box** – creates a box around all active layout geometry across all layers, aligned with the X/Y axes.
        - **Bounding Box (oriented)** – the bounding box is oriented around all active layout geometry across all layers, rather than aligning with the XY axes, to create the tightest bounding box possible.
    b. Enter an appropriate amount of padding for each side of the object in the X, Y, and Z fields.
    c. If applicable, check the **Sync** boxes to match the padding on each side of the component.
5. From the **Initial Mesh Method** area, choose a meshing method. Tau is a beta feature specifically for use within a **Mesh Fusion** framework, a hybrid meshing technology that

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

combines tetrahedral and prism-based elements to support mixed-solver workflows. Refer to Activating the Tau Mesher Beta Feature

6. If necessary, check the **Auto-resolve domain intersections** box. By default, mesh fusion only allows dielectrics from the native domain to intersect with child domains. The intersecting dielectrics are removed from the envelope region completely. When **Auto-resolve domain intersections** is checked, the intersection is resolved and the geometry description is preserved. The dielectric and conductor bodies overlapping the Mesh Fusion envelope are split and moved to the mesh fusion domain.
7. Click **Apply** to preview the result in the **Layout Editor**. A black box is drawn around the object to denote the bounding box (envelope).
8. To make the mesh finer or coarser, click the **Curved Surface** tab. For a finer degree of control, uncheck the **Use dynamic surface resolution** box (refer to Surface Approximation).
9. To set the **Model Resolution**, click the **Advanced** tab. This sets the optimal length of the hypotenuse for triangles in a mesh (refer to Model Resolution Length).
10. To make the current selections the new default configuration, check the **Save as default** box.
11. Click **OK** to set the mesh assembly.

The procedure is complete.

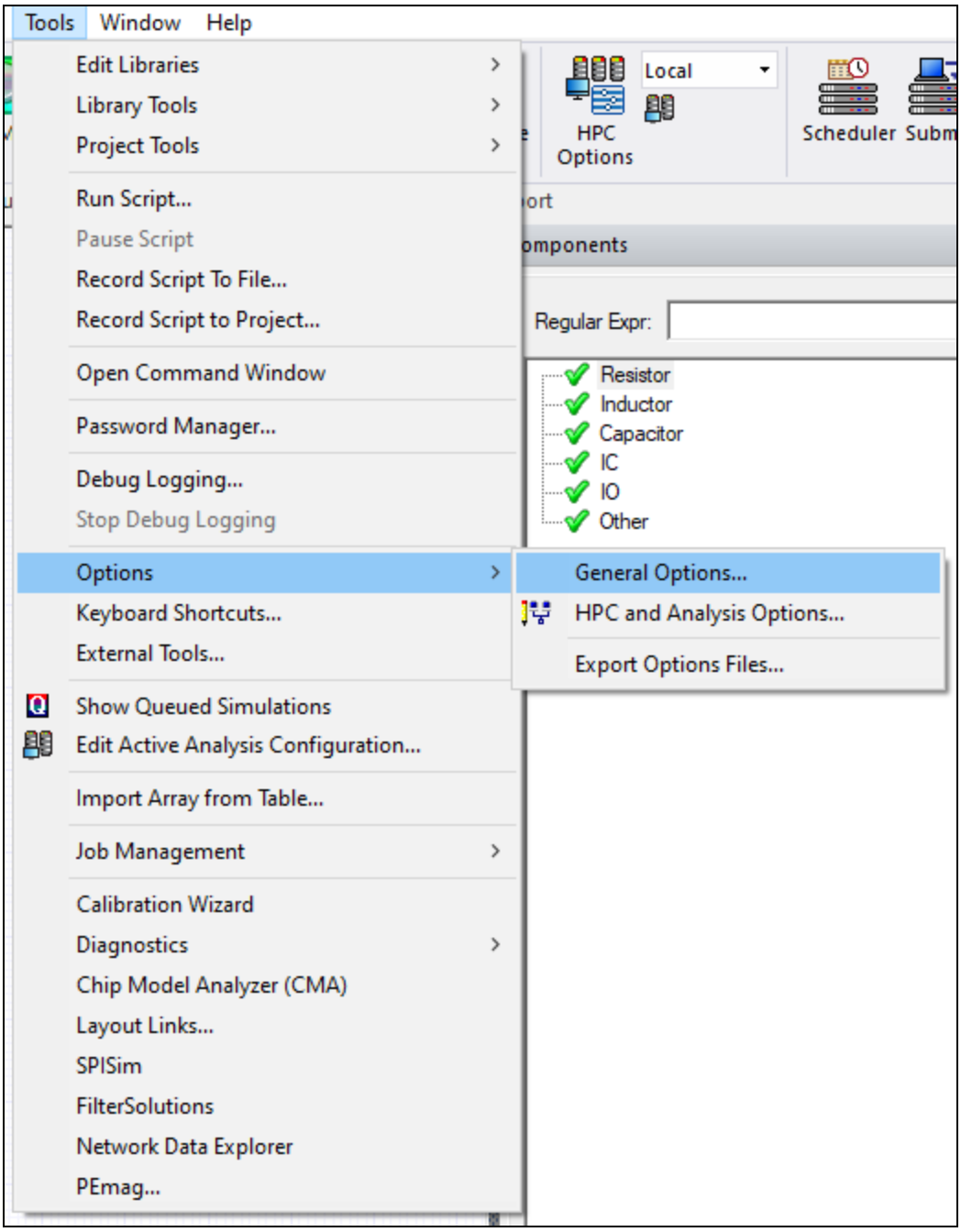
> **Note:**
>
> The 3D Layout mesh settings define settings for individual components and add the possibility of a mesh assembly envelope. ECAD components can only have a bounding box envelope. MCAD components can have bounding boxes and definition envelopes, given by the pre-defined Mesh Regions in the component. Refer to "Mesh Fusion/Assembly for Hierarchical Design" on page 7-390 for details.

Activating the Tau Mesher Beta Feature

Follow these steps to enable the Tau mesher beta feature.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

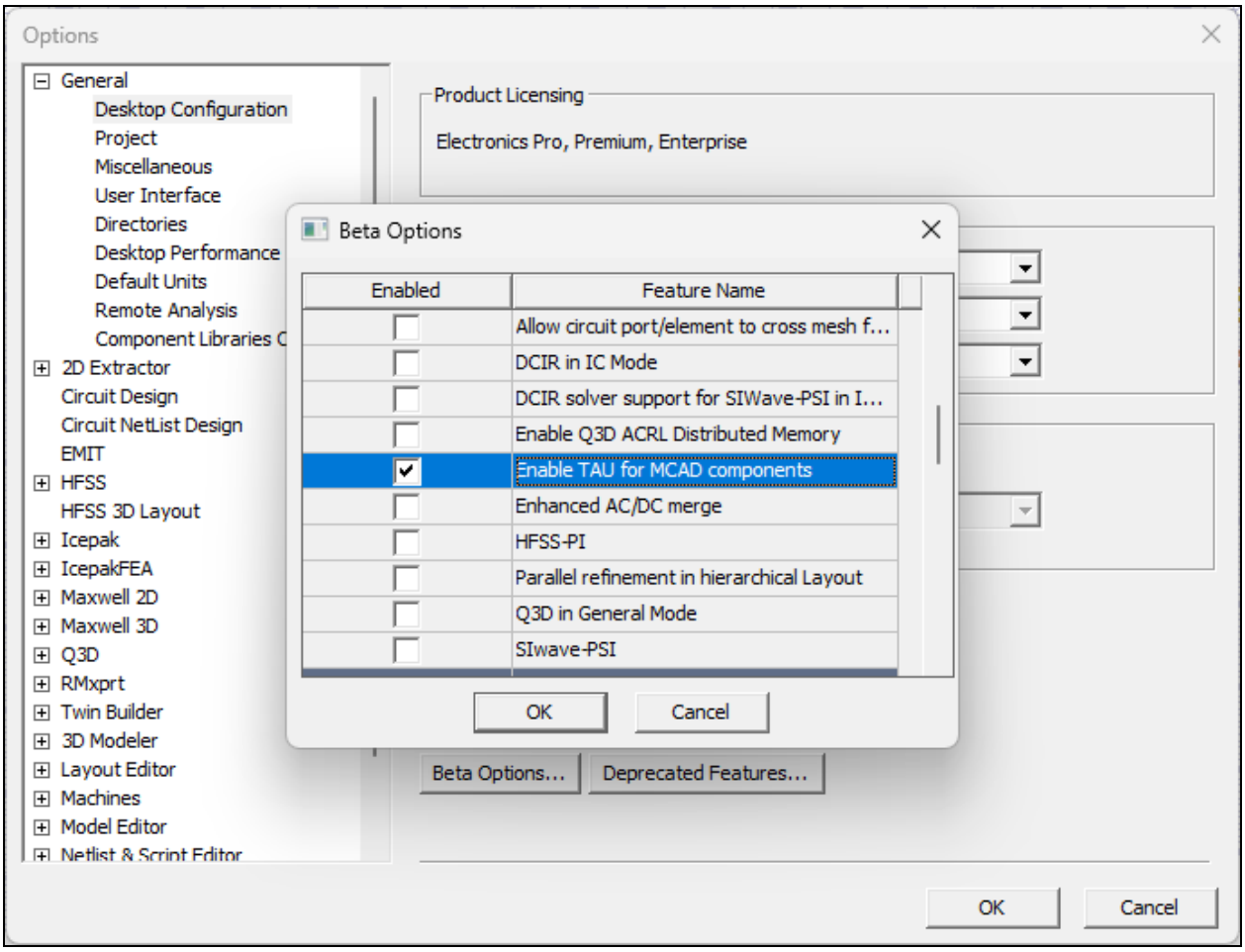
1. From the **Tools** menu, select **Options** > **General Options** to open the **Options** window



2. Expand **General** > **Desktop Configuration** and click **Beta Options** to open the **Beta Options** window

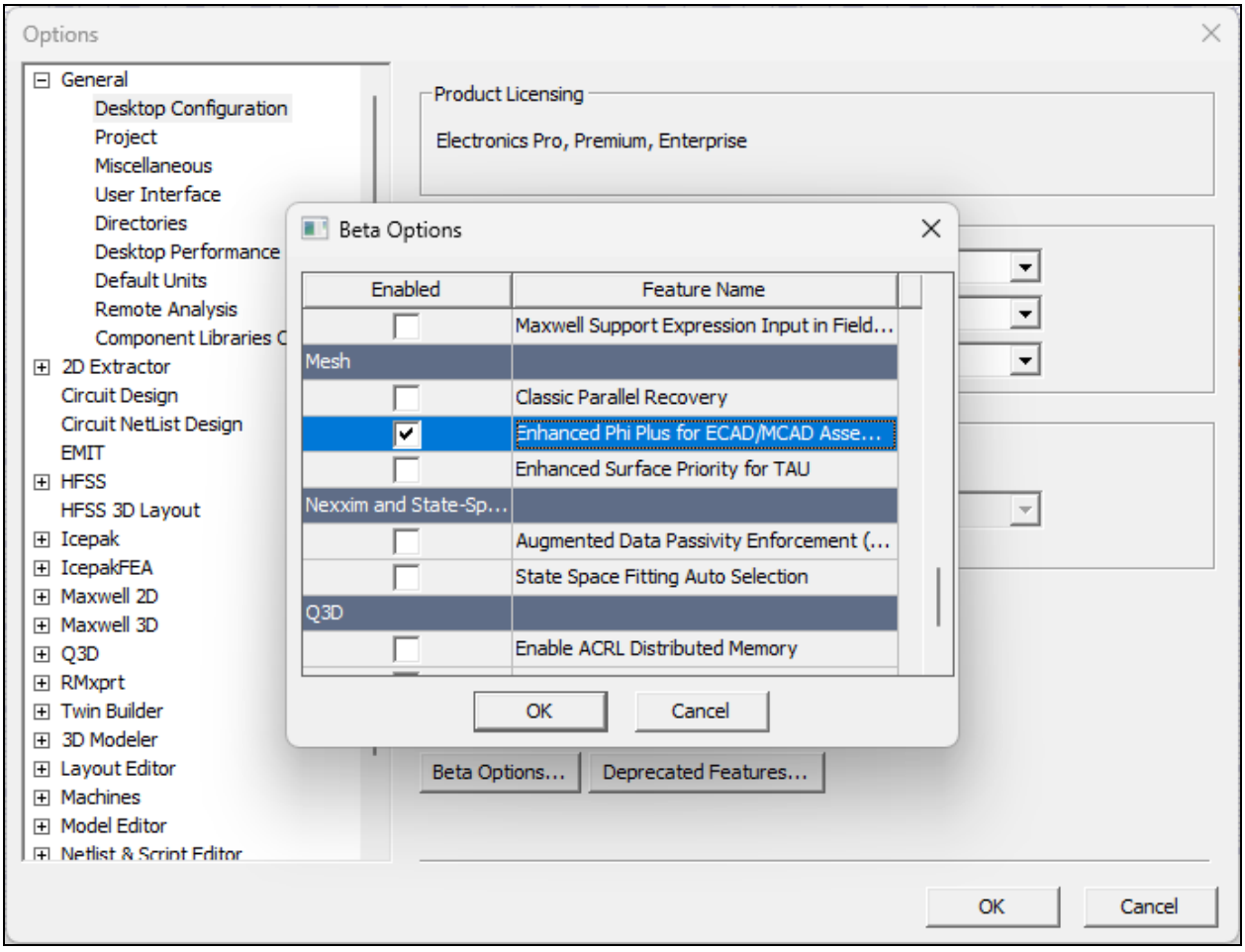
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scroll down to the **HFSS 3D Layout** section and check the box adjacent to **Enable TAU for MCAD components**.

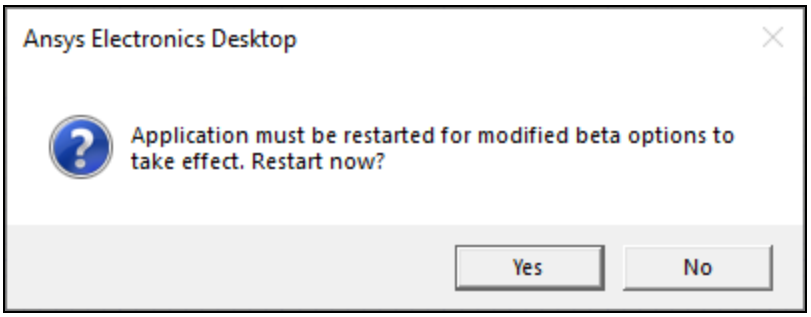


4. Scroll down to the **Mesh** section and check the box adjacent to **Enhanced Phi Plus for ECAD/MCAD Assembly**.

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 close the **Beta Options** window.
6. Click **OK** to close the **Options** window and open a prompt to restart the software. Users may be asked to save their work.



7. Click **Yes** to restart immediately.

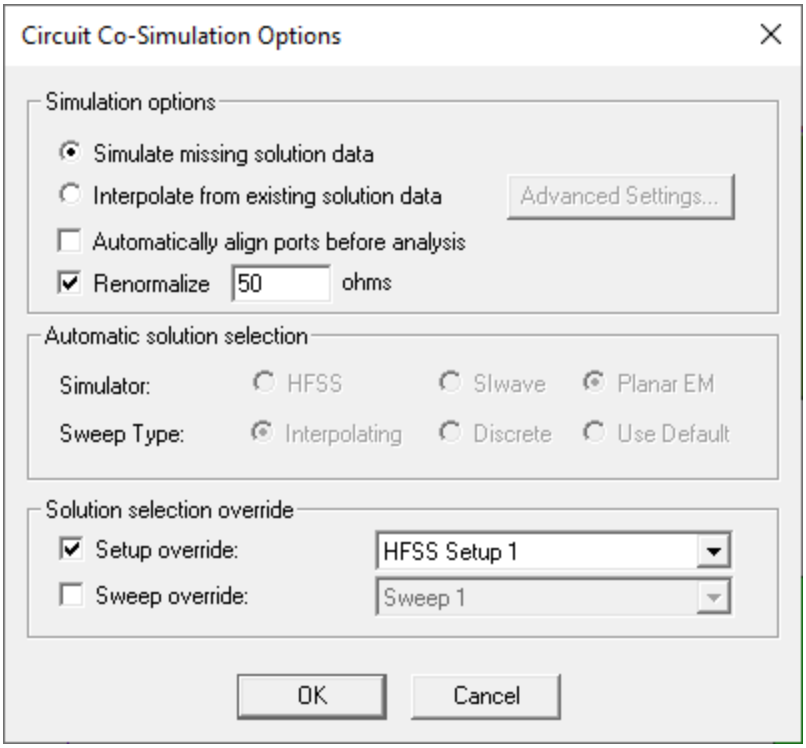
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. After restarting the software, the Tau mesher is available from the location described in the previous section.

The procedure is complete.

Pre-Processing Settings for Components

For component instances, pre-processing settings are derived on the design's Co-simulation Options. Specifically, to explicitly control defeaturing settings, arc resolution, and other settings that control geometry realization, define a selected HFSS Setup and assign it as the **Setup override** in the **Co-Simulation Options** window. To reach this window, click **HFSS 3D Layout** > **Solution Setup** > **Co-Simulation Options**.



Creating Mesh Regions and Activating Parallel Refinement (IC Mode Only)

Defining mesh regions in an HFSS solution setup partitions a design into multiple domains, potentially increasing mesh robustness and performance for complex layout geometry. For larger designs, enabling mesh regions also allows users to activate parallel refinement, a process that significantly accelerates adaptive passes by refining each region individually to reach a final solution.

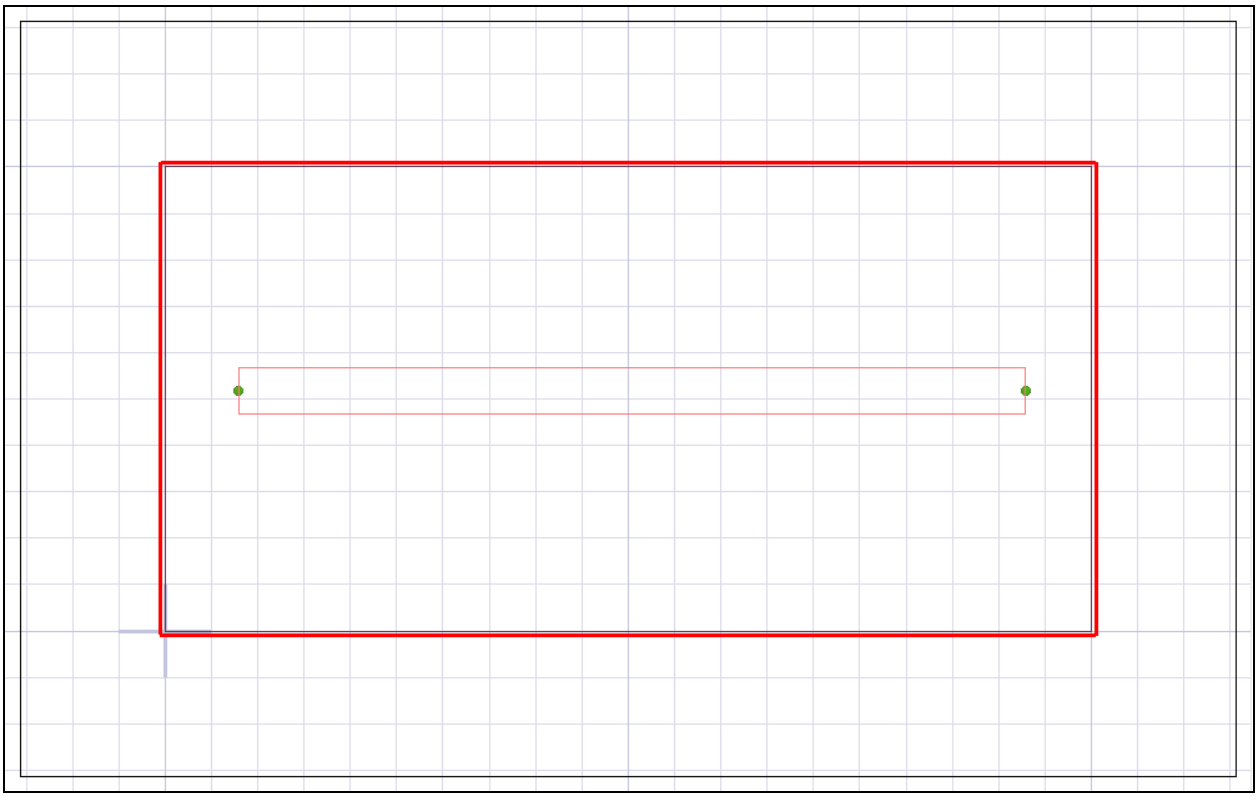
Follow these steps to create a mesh region and configure mesh partitions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

These instructions assume the user has already set up an Advanced HFSS Solution Setup.

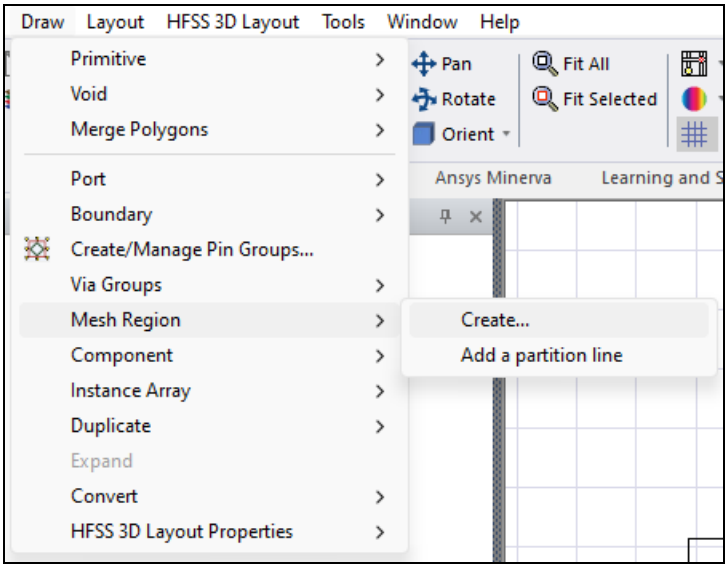
1. From the **Layout Editor**, select a component to designate as a mesh region.



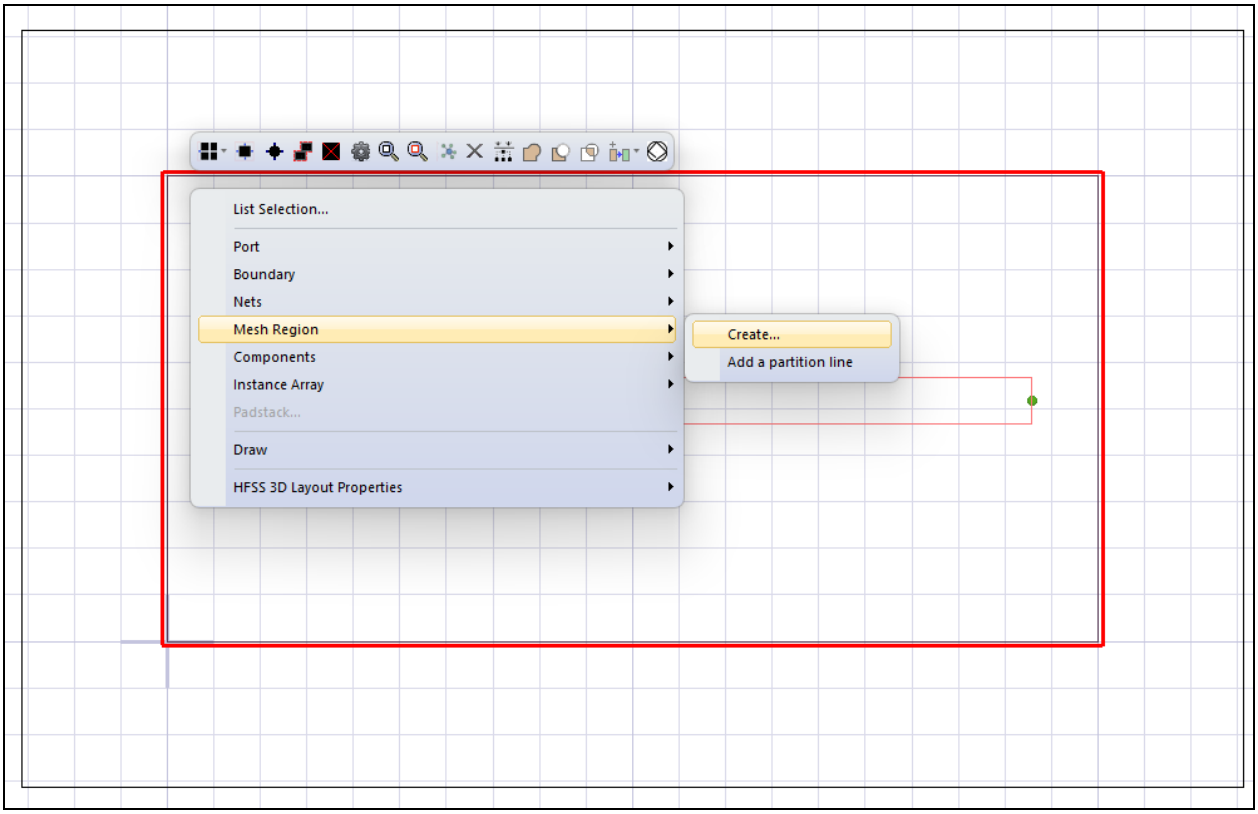
2. Do either of the following to open the **Mesh Region Dialog** window:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From **Draw**, select **Mesh Region** > **Create**.

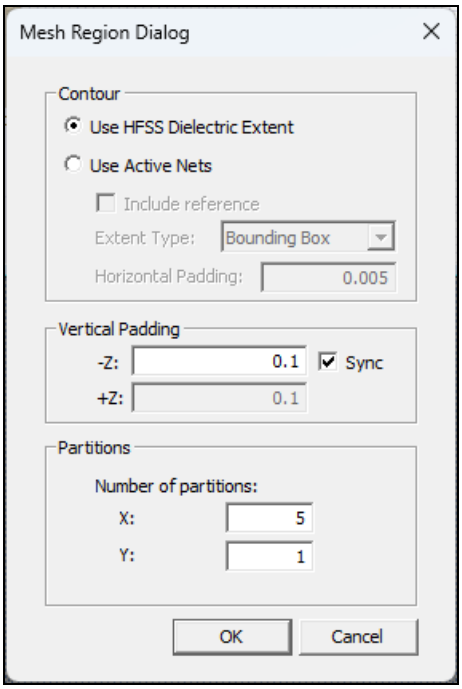


- Right-click within the **Layout Editor** and select **Mesh Region** > **Create**.



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 **Mesh Region Dialog** window, make any other appropriate selections and enter values in the **Partitions** area.



Note:

By default, mesh regions created in IC mode always extend to the airbox extent, regardless of the selection made in the **Contour** area of the **Mesh Region Dialog** window. This simplifies mesh boundaries and improves accuracy. Selections made in the **Contour** area only become relevant when an IC mode design is exported and later imported into General mode as a component. Once imported into General Mode, the **Contour** selections are respected.

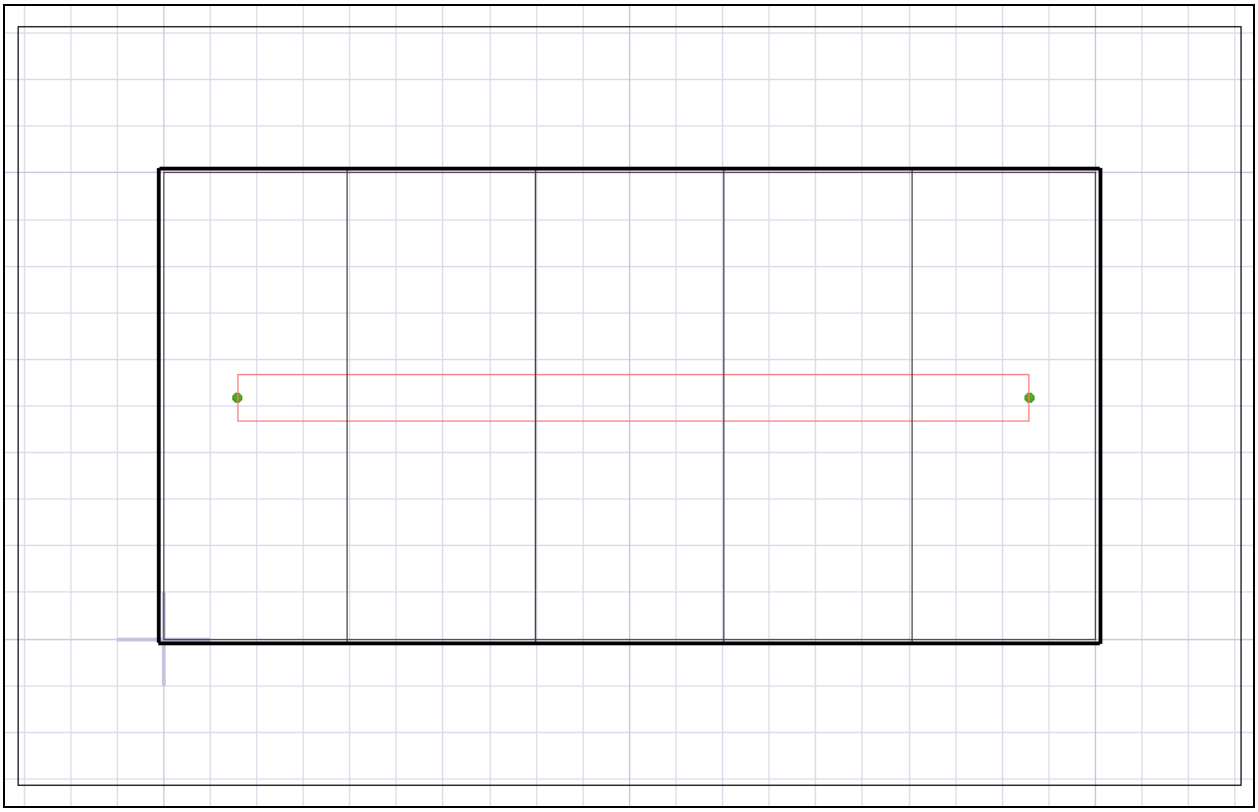
Note:

The default **X** and **Y** partition values depend on the length of the bounding box around the objects in the **Layout Editor**. If the X-axis side is longer, the **Number of partitions** is **5** in the **X** field and **1** in the **Y** field. If the Y-axis side is longer, the **Number of partitions** is **1** in the **X** field and **5** in the **Y** field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

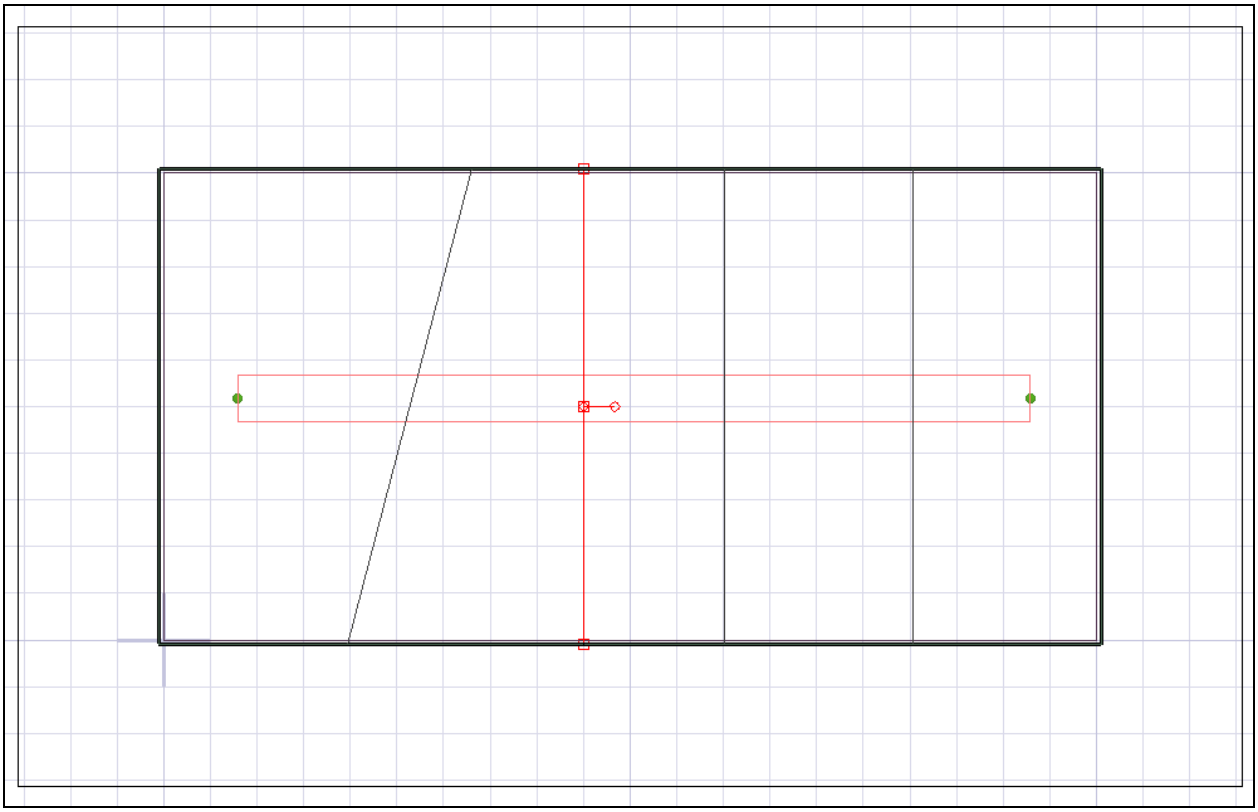
> **Note:**
>
> By default, both X and Y axes (i.e., 2D) partitions are editable. To use only one axis (i.e., 1D) partition, enter **0** in the superfluous field.

4. Click **OK** to close the **Mesh Region Dialog** window and the partitions will appear in the **Layout Editor**.



5. **Click+drag** on partition lines to drag them or select a line and use the handles that appear to transform the partition(s), as necessary.

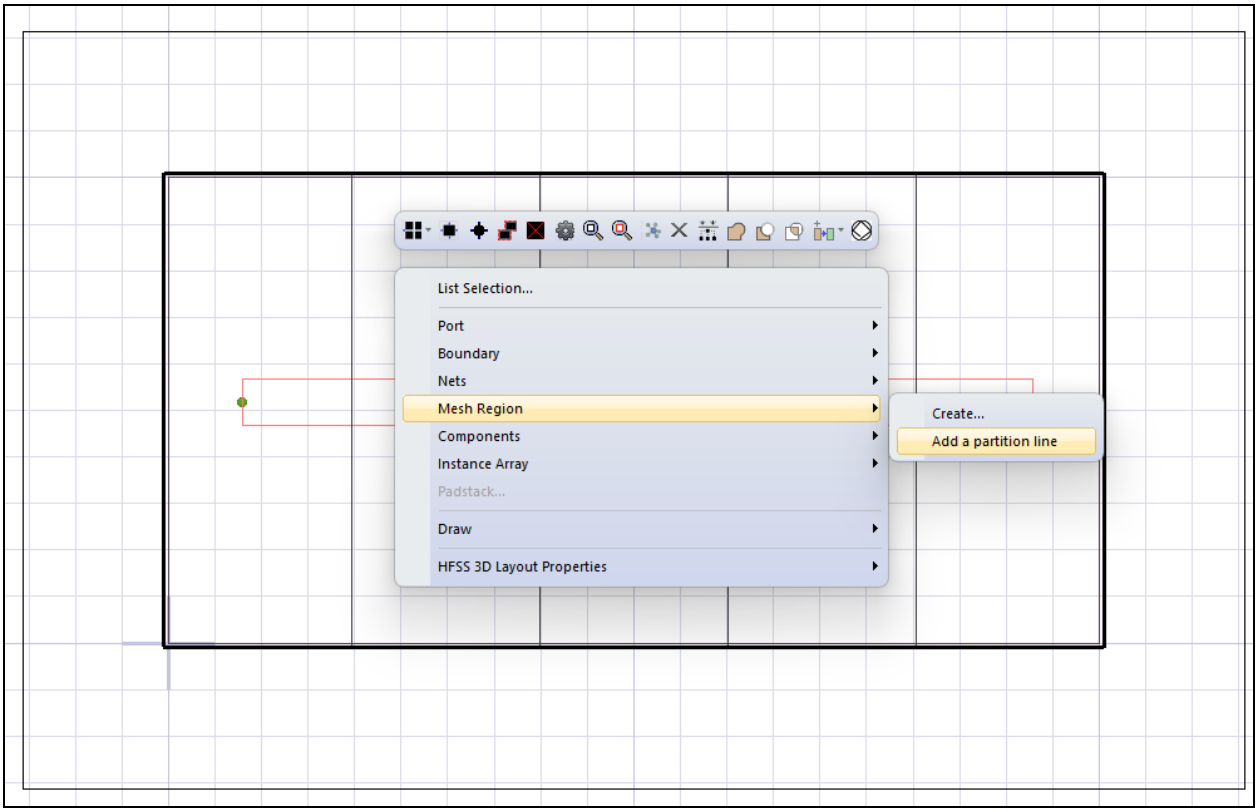
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. If necessary, add a new partition line by doing the following:

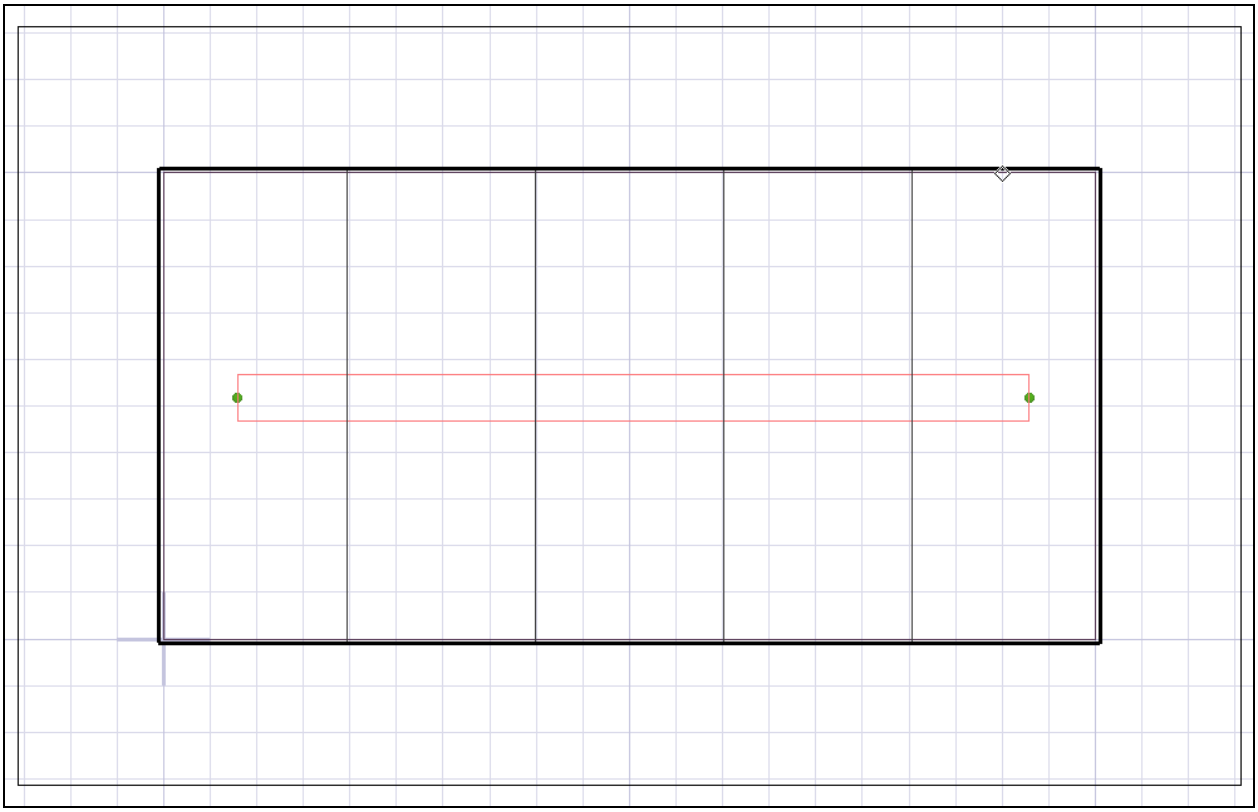
    a. From the **Layout Editor**, select the mesh region. Then right-click and select **Mesh Region** > **Add a partition line**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



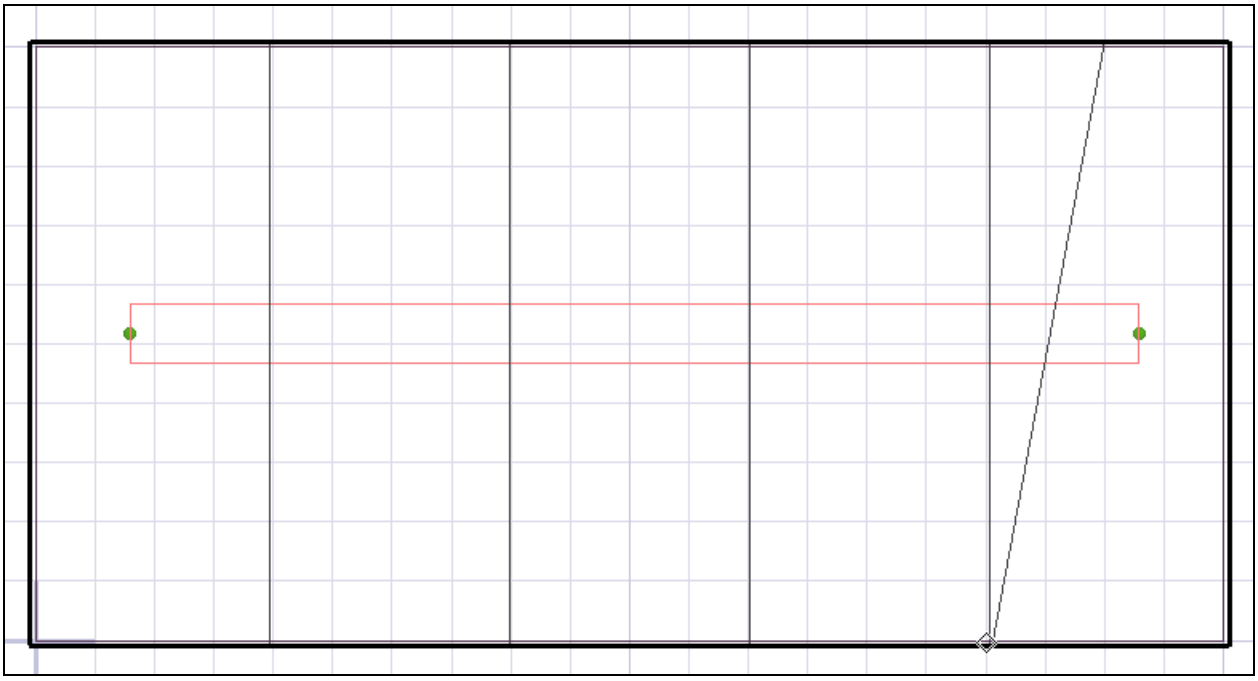
b. Click anywhere along the mesh region outline to start a new line.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



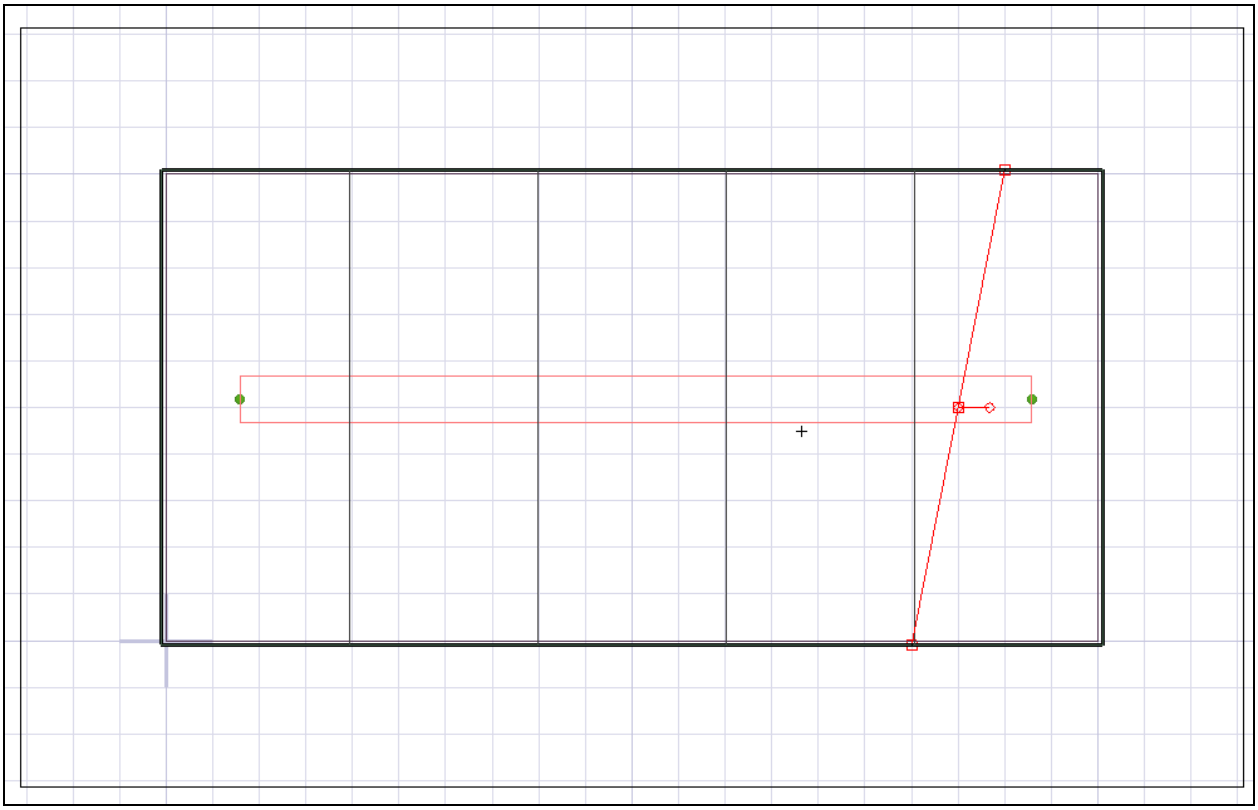
c. Drag the mouse to another side of the mesh region outline and click again to see the new line segment.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



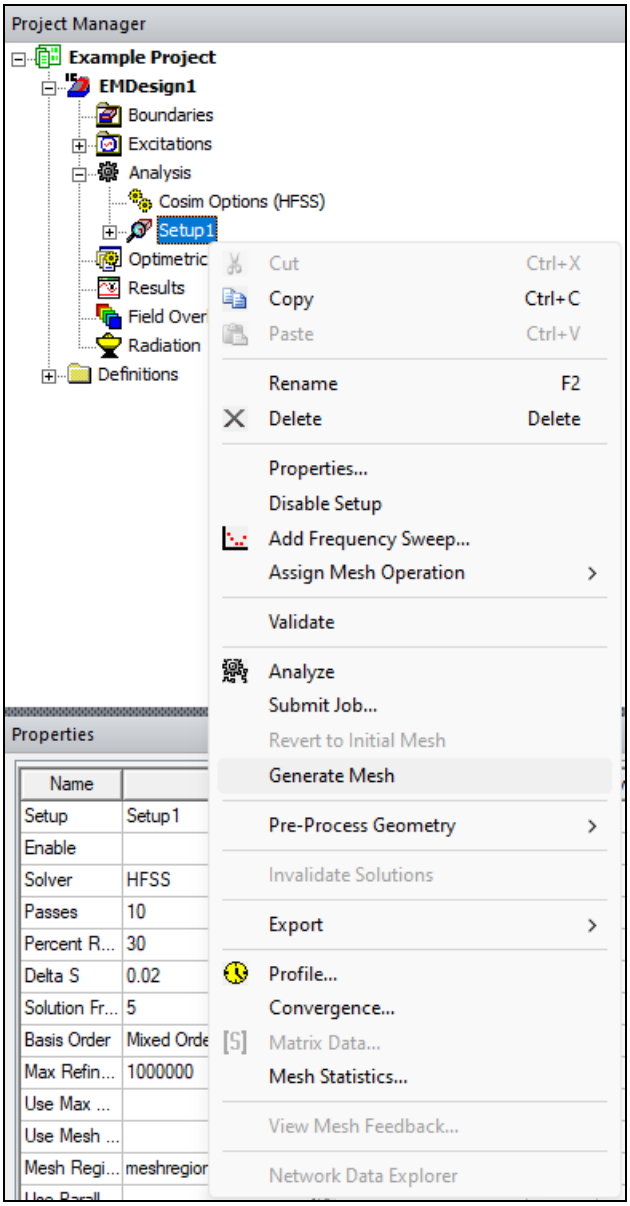
d. Repeat step **c**, as necessary, to create more line segments, or press **Enter** to create the new partition line.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



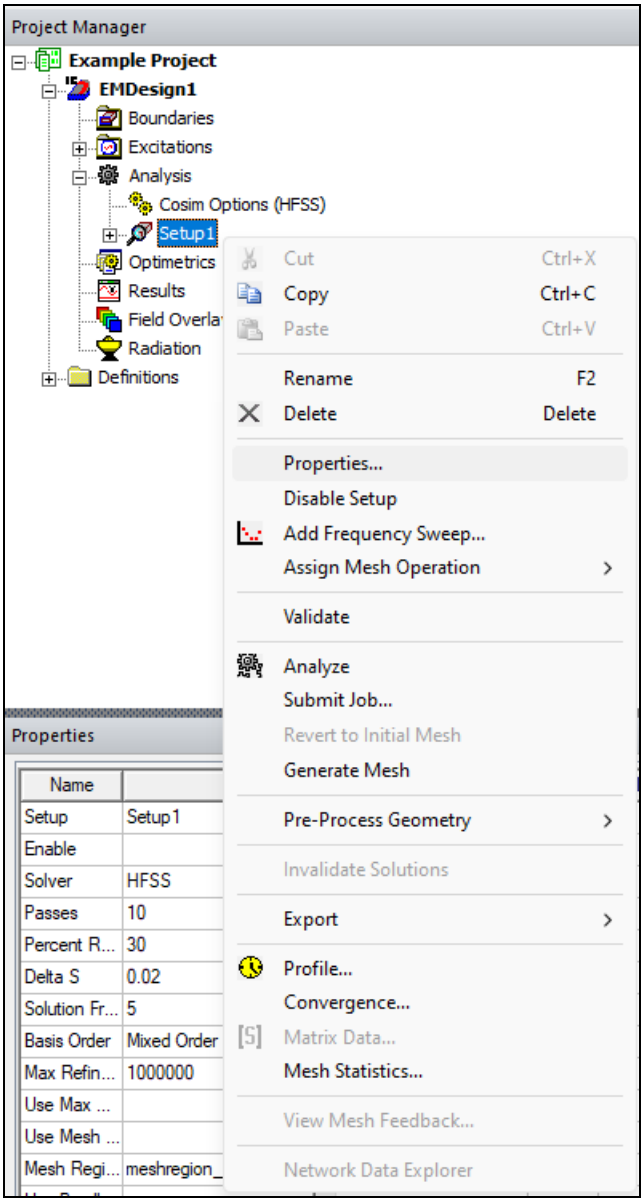
7. When the mesh region is complete, navigate to the **Project Manager** window. Expand the **Project Tree** > **Active Design Folder** (e.g., **EMDesign1**) > **Analysis**. Then right-click the **Setup*n*** and select **Generate Mesh**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



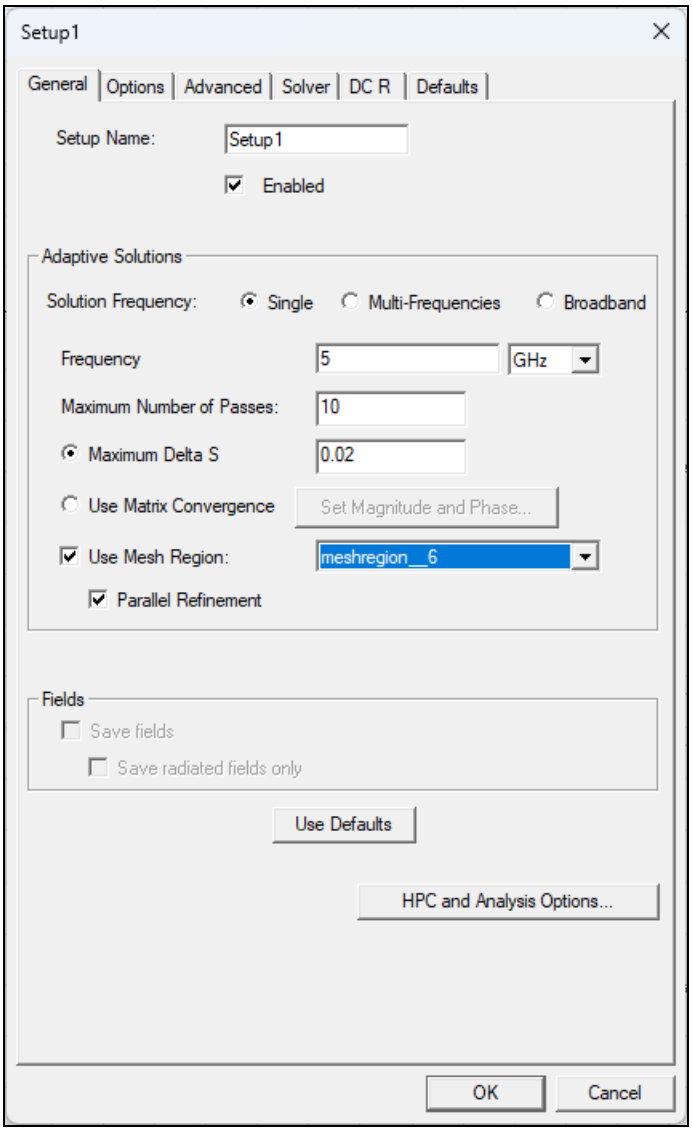
8. From the **Project Manager** window, right-click **Setup*n*** and select **Properties**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. From the **Setup** window, navigate to the **Adaptive Solutions** area and click **Use Mesh Region** to activate the drop-down menu and the **Parallel Refinement** 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.



10. Select the appropriate mesh from the drop-down menu (e.g., **meshregion_6**).
11. By default, mesh regions are meshed in parallel but then stitched together and adaptively refined as a single domain. Checking the **Parallel Refinement** box allows each partition to be refined simultaneously, significantly reducing adaptive meshing time for complex designs. This trade-off prioritizes faster simulation while maintaining overall solution quality.

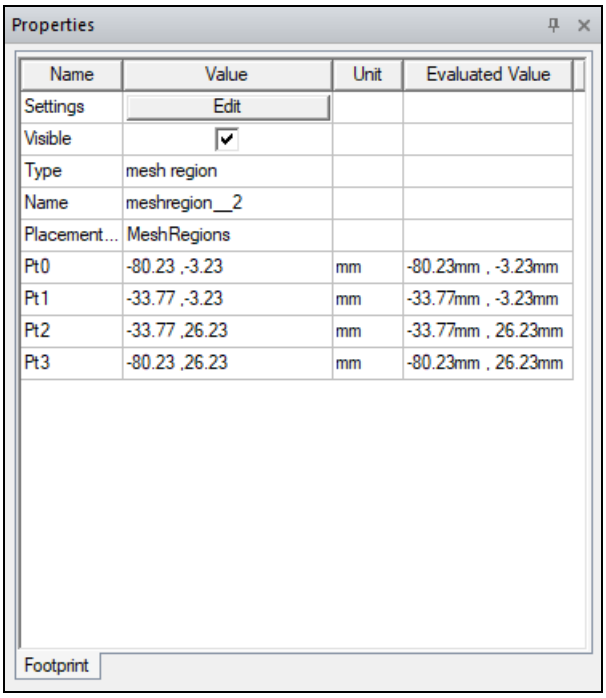
The procedure is complete.

Editing the Mesh Region

Once the mesh region is created, edit the parameters or edit/reset the number of partitions from the **Properties** window. Follow these steps to reset the number of partitions.

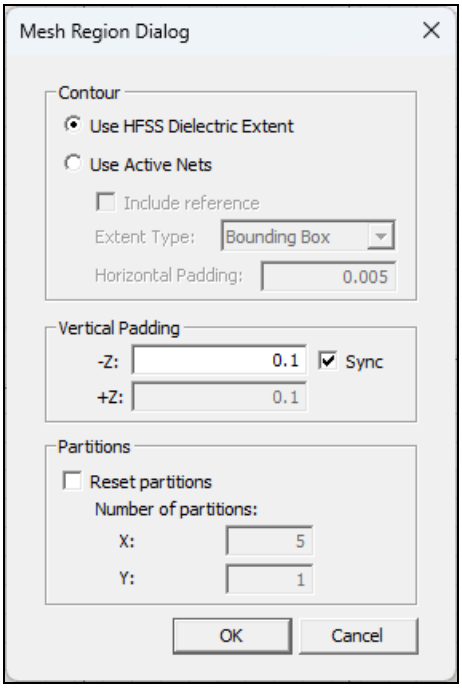
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click on the mesh region outline to simultaneously select the mesh region and populate the **Properties** window with the mesh region's parameters.

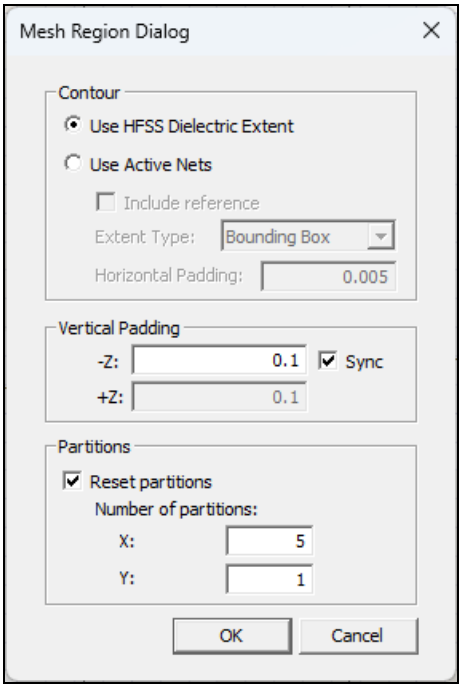


2. Click **Edit** to reopen the **Mesh Region Dialog** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Check the **Reset partitions** box and click **OK** to reset the **Number of partitions**. The **Layout Editor** automatically updates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

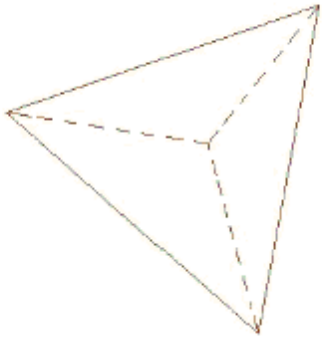
Mesh Generation

Mesh generation is the process by which space-discretization is implemented. During mesh generation, a geometric domain is divided into a number of elements.

The type of the mesh is defined by the shape of the element, which can be tetrahedral, hexahedral, polyhedral, pyramidal, prismatic etc.

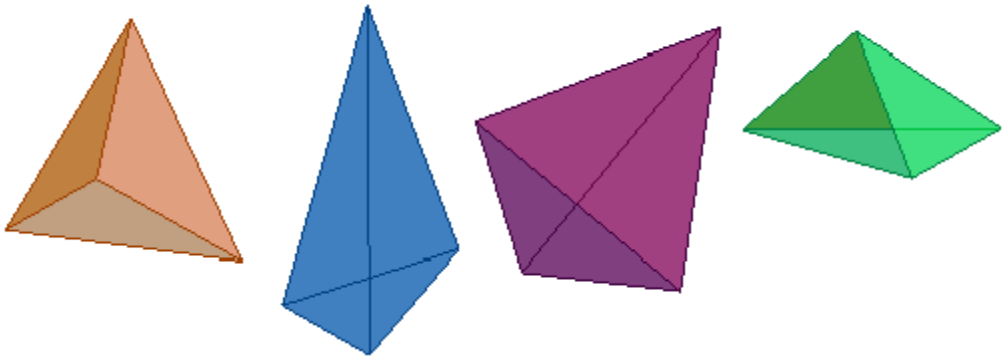
In HFSS, a mesh is a discretization of a continuous domain (geometry) into small three-dimensional tetrahedral and two-dimensional triangular elements. The following figure shows a tetrahedron, which is a solid composed of four triangular faces.

A tetrahedron



Tetrahedra are the simplex for R3 space, which means that any 3D shape can be decomposed into tetrahedra. Since the process of mesh generation is automatic in HFSS, this mathematical guarantee is necessary for successful mesh generation of arbitrary domains. Therefore tetrahedra (and triangles for two-dimensional surfaces) are used. Tetrahedra can be stretched and pulled (see the figure below), which allows them to fit any arbitrary 3D geometry; so any 3D shape can be decomposed into a number of tetrahedra.

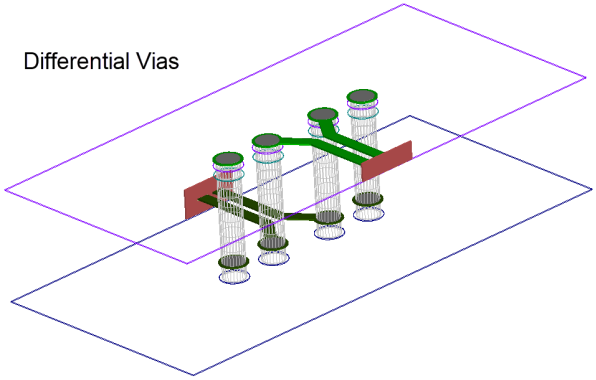
Different kinds of tetrahedra



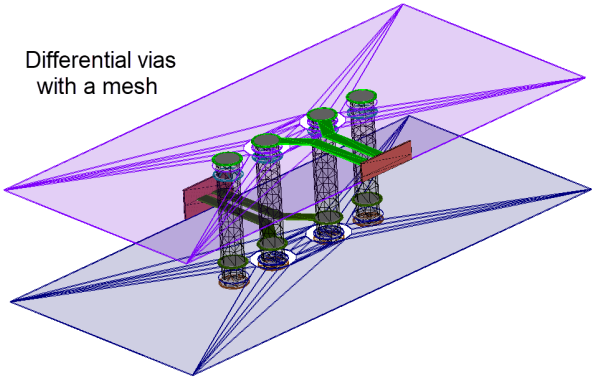
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figures illustrate HFSS 3D Layout designs of differential vias before and after the mesh is created. Notice that the model is well-represented by the mesh.

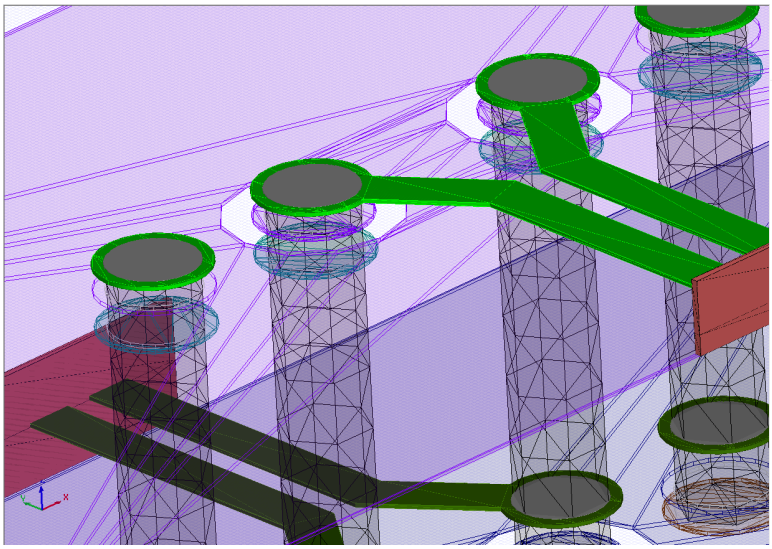
Differential Vias



Differential vias with a mesh



Close up of the vias



Generating Mesh without Solving

If you want to refine the mesh on a face or volume but do not want to generate a solution, define mesh operations. Then do one of the following:

- Select [**HFSS 3D Layout**] > **Analysis Setup** > **Generate Mesh**.
- Right-click the **Analysis** or **Setup** icon in the **Project** window and select **Generate Mesh**.

The same solve machine rules that apply to solving any other setup also apply here.

- If a current mesh has been generated, the solver will refine it using defined mesh operations.
- If a current mesh has not been generated, the solver will apply the mesh operations to the initial mesh.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If an initial mesh has not been generated, the solver will generate it and apply the mesh operations to the initial mesh.

Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them or by reverting to the initial mesh.

Adaptive Mesh Refinement

Adaptive mesh refinement automatically generates accurate solutions. This meshing technique helps you focus on setting up your design efficiently rather than spending time in determining and creating the best mesh. To set up the design, you need only to create the geometry and specify material properties, boundary conditions, excitations, and the solution frequency.

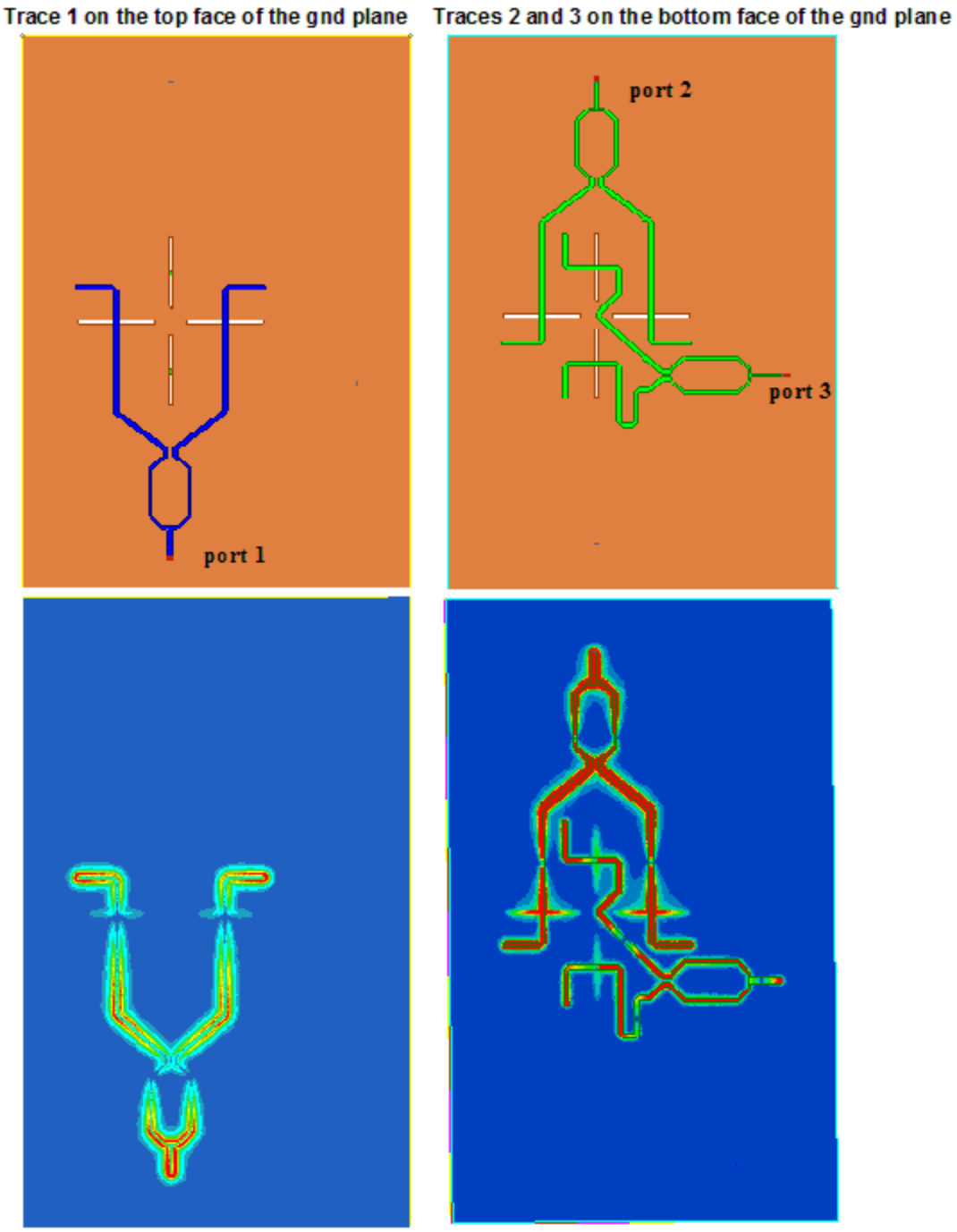
This automatic adaptive mesh refinement technique differentiates HFSS from most electromagnetic simulators, which typically require extensive user controls to ensure that the generated mesh is suitable for simulation. Without the correct mesh, results from such simulators may be erroneous.

In HFSS, however, the automatically generated mesh accurately represents the electromagnetic characteristics of the design. HFSS determines the size and density of tetrahedra for optimal efficiency and accuracy. An initial mesh is generated and solved to produce error estimates of the solution. These error estimates are used to refine the mesh in high-error regions. Using this refined mesh, HFSS generates another solution and recomputes the error. This adaptive refinement process continues until HFSS converges to an accurate solution. Convergence is determined by monitoring a parameter from one adaptive pass to the next. The most common convergence criterion is to ensure that the difference in the S-parameter value between two consecutive solves is less than the specified magnitude. If you need to increase the accuracy, you can easily tighten the convergence criteria, and HFSS further refines the mesh.

Adaptive refinement ensures that the mesh elements are sufficiently fine in those areas where strong electromagnetic fields exist and/or the field gradients are high. The mesh is coarser in the remaining areas. This process makes the mesh size appropriate and suitable for efficient simulations leading to highly accurate solutions.

To illustrate, an aperture coupled microstrip filter design with three traces is shown below. One of the traces is located on one side of the ground plane, while the other two traces are located on the opposite side. The figures also show the resulting electric fields that develop for the design after the simulation is completed. The plots show the traces of the microstrip on both sides of the ground plane illuminated by the electromagnetic fields.

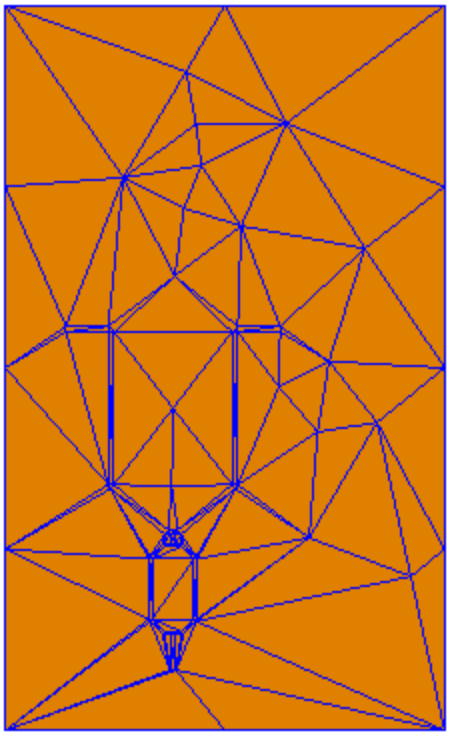
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



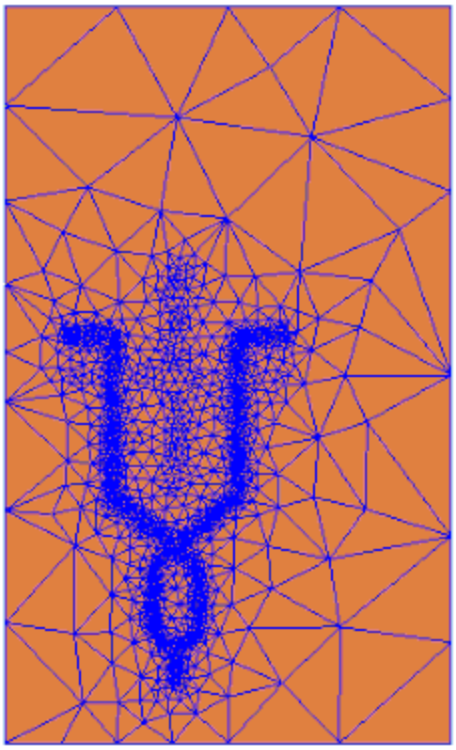
See both the initial and final meshes for these two layers and how they are automatically adapted by the adaptive refinement algorithm.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

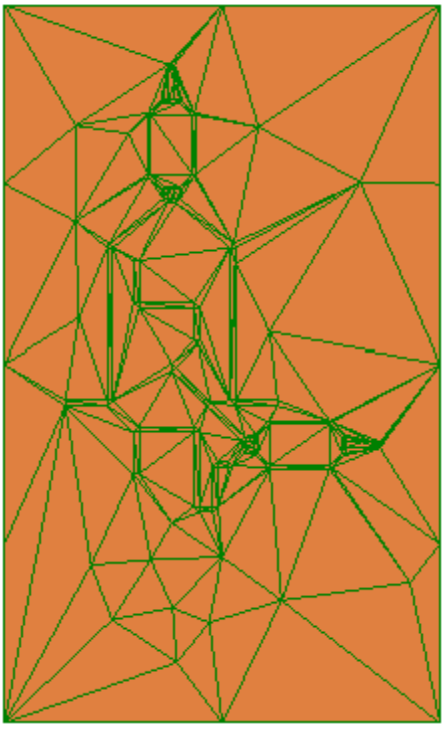
Initial mesh on the top face of the gnd plane



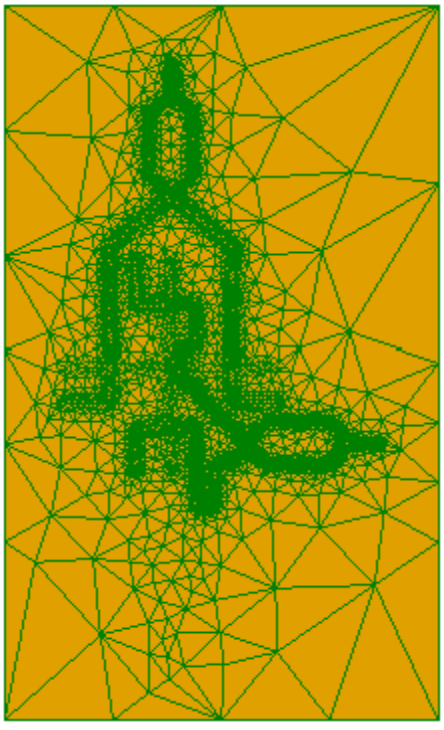
Final mesh on the top face of the gnd plane



Initial mesh on the bottom face of the gnd plane



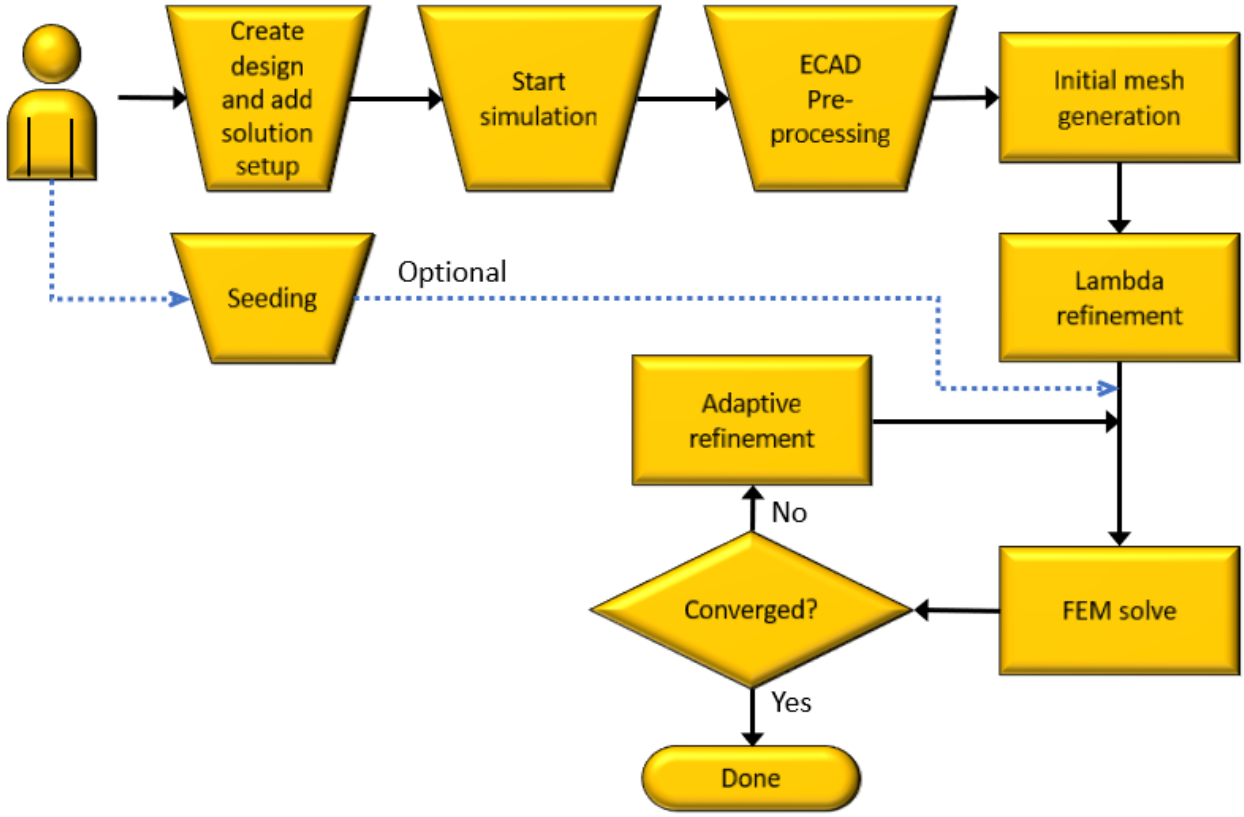
Final mesh on the bottom face of the gnd plane



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adaptive Mesh Refinement Process Flow

HFSS uses the adaptive mesh refinement technique and generates accurate solutions with minimal use of computational resources. The following diagram shows the adaptive mesh refinement process flow that can deliver accurate results with least user involvement.



For mesh operations, defining seeding is an optional step in the solution process, as shown in the flow diagram. This is because the automatic adaptive refinement process is adequate to produce accurate results. Only on rare occasions, adaptive refinement is used with mesh operations.

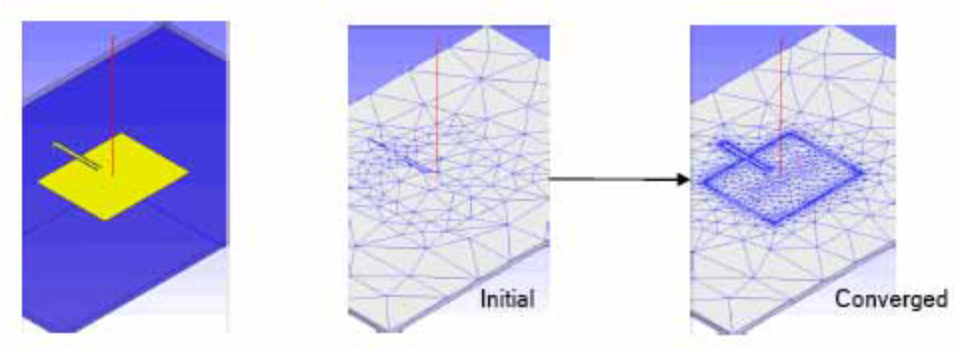
The process depicted by the flow diagram can be summarized as follows:

1. HFSS generates an initial mesh that can be controlled by surface approximation settings.
2. Seed refinement (if assigned) and Lambda refinement occur on the initial mesh.
3. Using the resulting mesh, HFSS computes the electromagnetic fields in the structure.
4. From the current solution, HFSS produces error estimates and refines the mesh in regions of high error and uses the refined mesh to generate another solution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. HFSS performs the iterative process of solve, error analyses, and adaptive refinement until the convergence criteria are satisfied or the maximum number of adaptive passes is completed.

The following figure illustrates that the initial mesh is refined until convergence criteria are satisfied and a suitable final mesh is generated.



Stages of Meshing

During mesh generation, HFSS divides the geometric domain into tetrahedra. The collection of tetrahedra forms the finite element mesh. The different stages of meshing are described as follows:

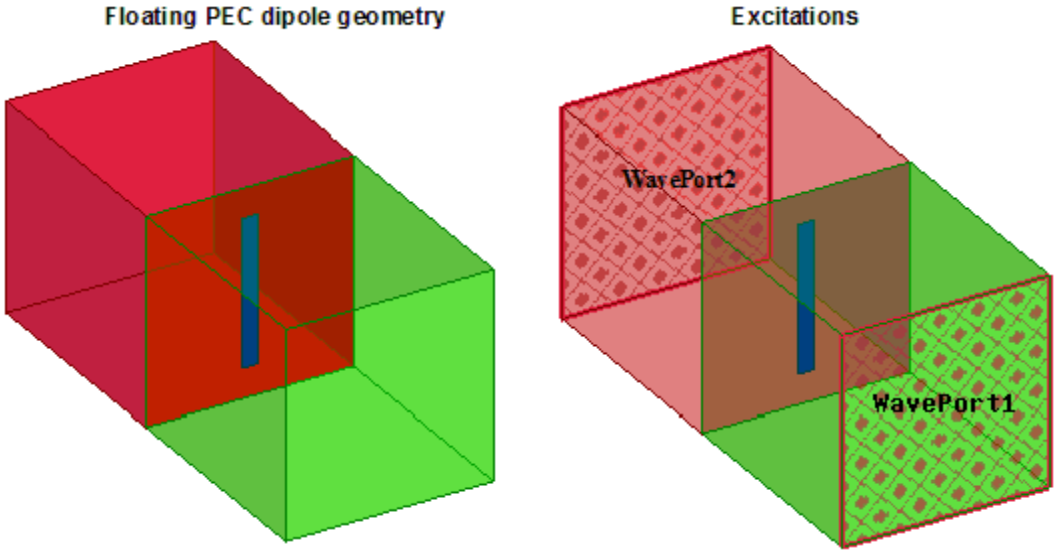
1. HFSS creates an initial mesh that can be controlled by various Setup parameters.
2. The initial mesh is refined based on the material-dependent wavelength lambda refinement and any mesh operations that are defined.
3. HFSS iteratively refines the 2D mesh at any ports in the model.
4. The electromagnetic fields are computed inside the structure.
5. The current solution is used to estimate the tetrahedra where the solution has large error and those tetrahedra are refined. This results in a refined mesh.
6. The refined mesh is used to generate another solution.

HFSS continues with the iterative process (steps 4and 6) until the convergence criteria are satisfied and then generates accurate results.

Example: Floating PEC Dipole

The following figures show a floating PEC dipole enclosed in a parallel plate waveguide excited at both ends by wave ports.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Although floating PEC dipole appears to be a simple design to solve, it is a canonical project for testing adaptive meshing algorithms. In this problem, the challenge is to efficiently refine the mesh for capturing the electromagnetic field singularities around the dipole.

The following figures illustrate how adaptive refinement addresses this challenge. First HFSS generates an initial mesh as shown in Figure 1. This initial mesh serves as a good starting point for adaptive refinement. Figure 2 shows the mesh plot after Adaptive Pass 1. This automatic adaptive refinement process continues until the convergence criterion is satisfied. Figure 3 shows an intermediate mesh during a later stage of the simulation. Figure 4 shows the final mesh that is produced after the solution converges.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

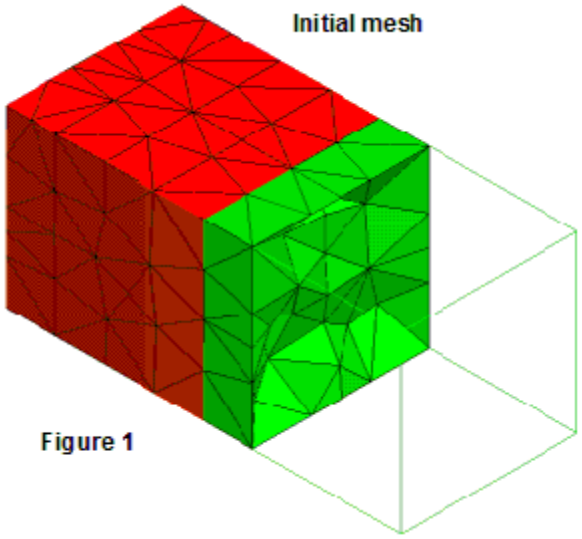


Figure 1

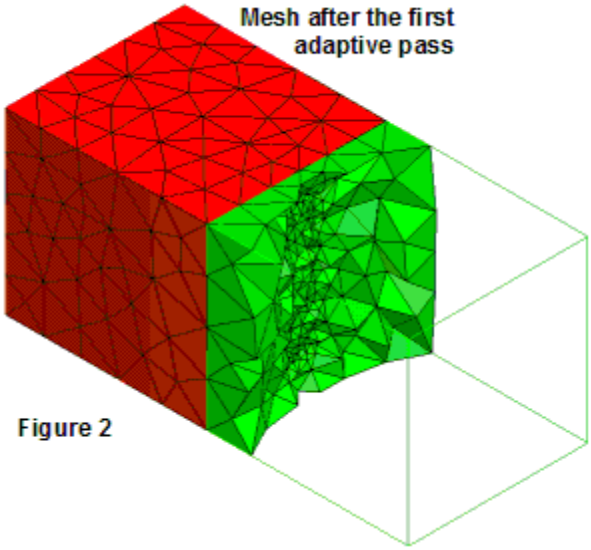


Figure 2

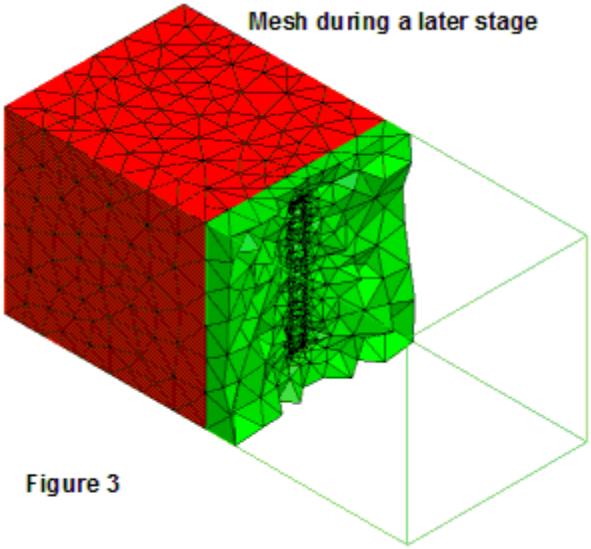


Figure 3

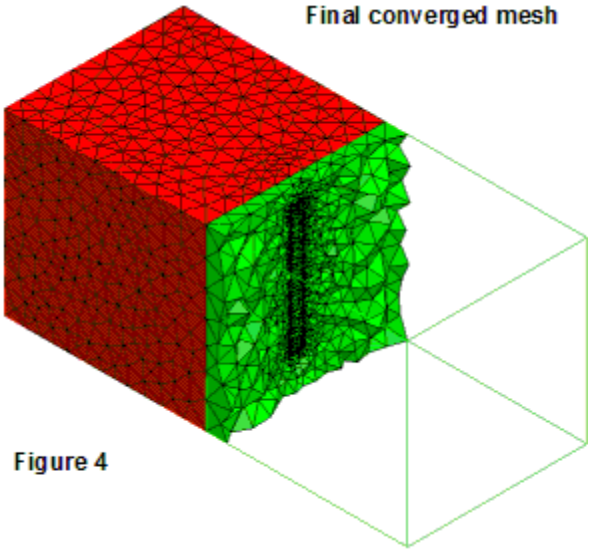
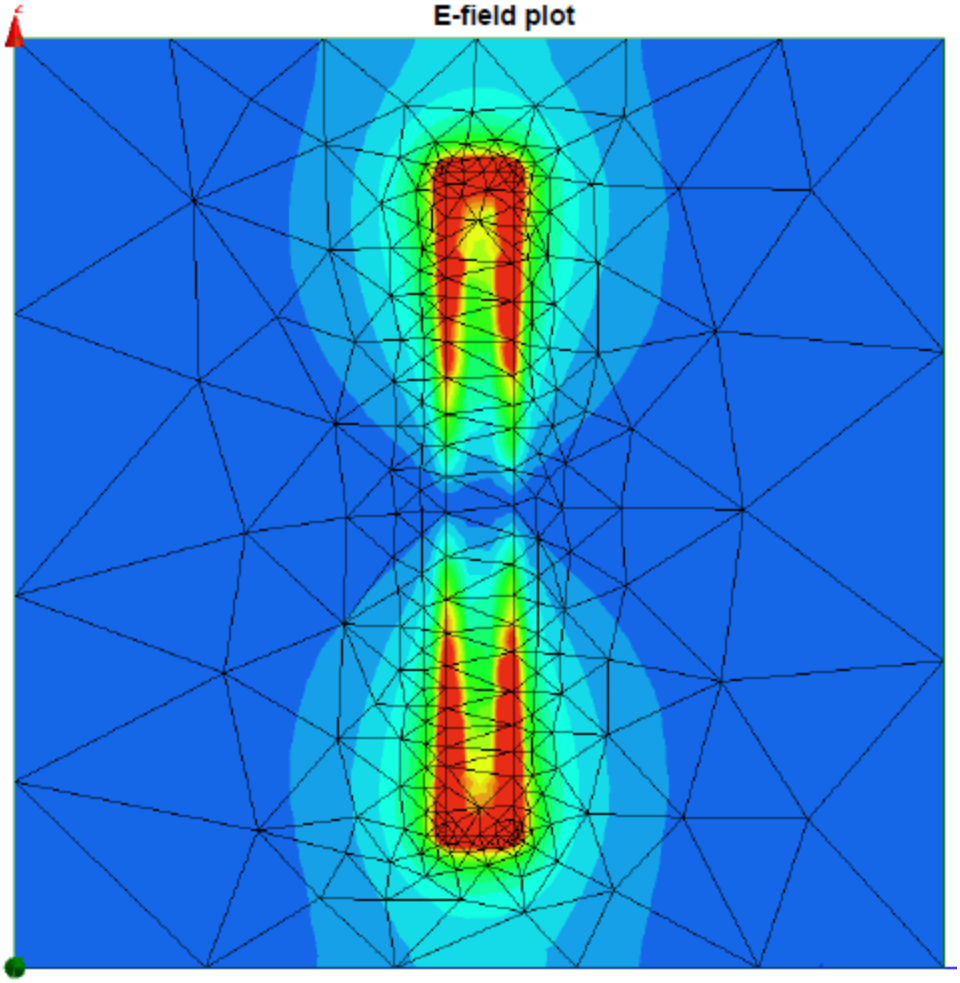


Figure 4

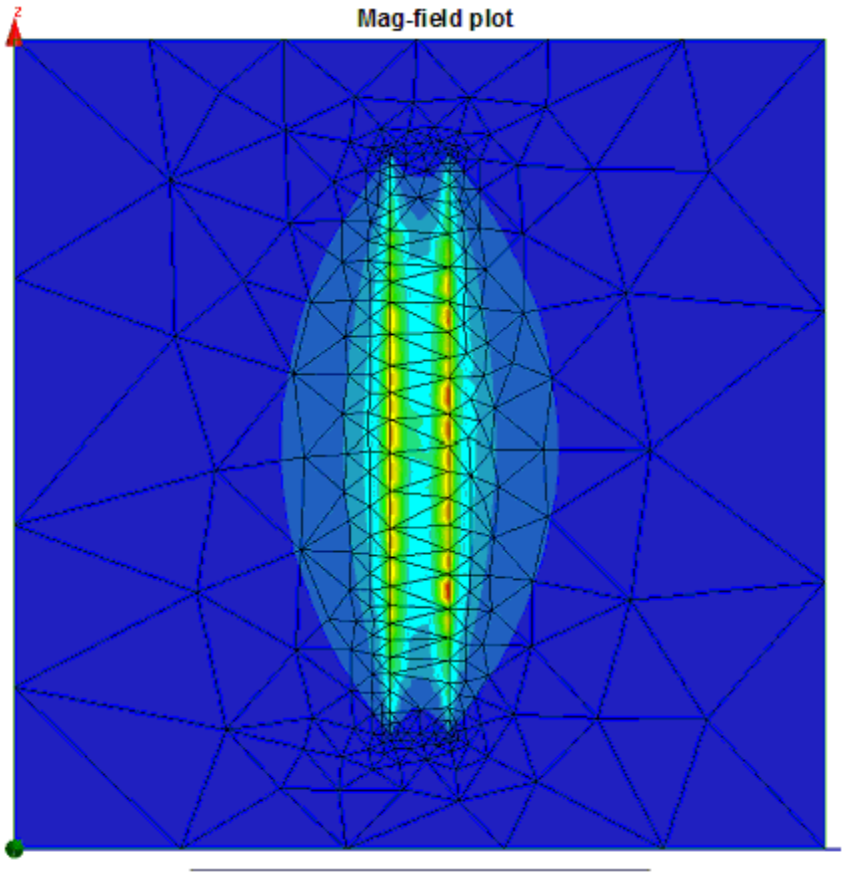
The following figure shows the electric fields with the mesh overlay.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



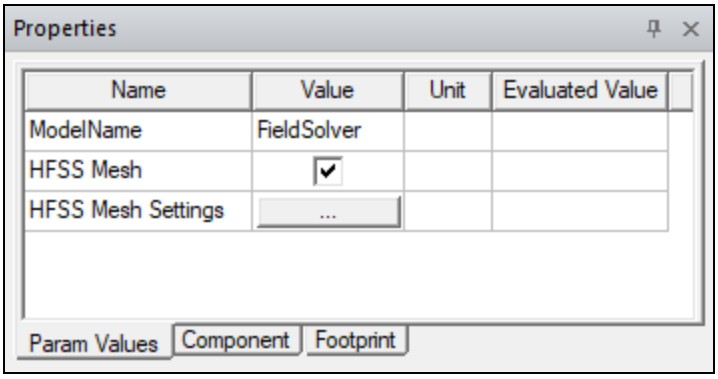
The mesh is fine around the center of the dipole, although the electric fields are weak, because strong magnetic fields exist in that region. This is shown in the following figure of the magnetic field plot on the same mesh.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Choose a Solver

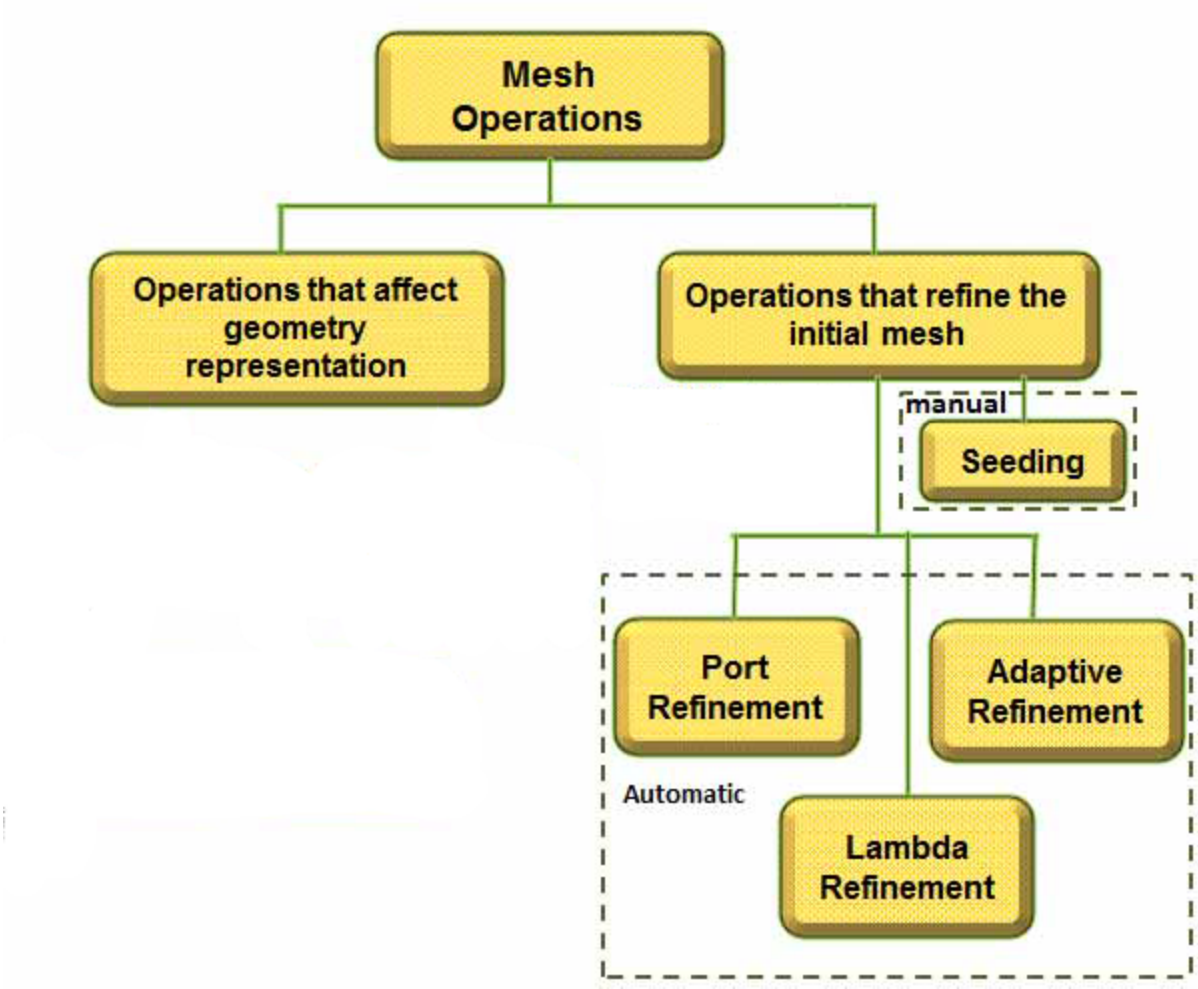
Sub-designs and 3D components can be analyzed and solved by the global HFSS finite element solver or independently by the Circuit co-simulation to integrate the results. To include the sub-design or 3D component in the global finite element solver, select the **HFSS Mesh** property for the sub-design. Because this option is for HFSS only, you must clear this check box if you intend to use the sub-design or 3D component with the Circuit co-simulation.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assign Mesh Operations

Mesh Operation settings are optional. The following figure provides a synopsis of the different mesh operations available in HFSS. The figure shows a classification only; it does not show the solution process flow. For more information about the solution process, see Stages of Meshing and Adaptive Refinement Process Flow.



Exert more control over the initial mesh when you want to accelerate resolution of specific electromagnetic effects; for example skin-depth, resonant structures, propagation characteristics for long transmission lines, etc.

Revert to Initial Mesh

For HFSS setups, if you do not want to use the existing current mesh, you can revert to the initial mesh before starting a solve.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reverting to the initial mesh deletes all solution data for a solve setup and all of its sweeps. Only the initial mesh information is saved. The next time a solve is started, it starts on the lambda mesh refinement.

> **Note:**
>
> This feature is the same as the initial mesh feature in HFSS, except that it is not available for all setups at once. You can revert to initial mesh for HFSS setups one at a time.

Create Mesh Plot

From the **Project Manager** window, expand the **Project Tree** > **[*Active Design Folder*]** > **Field Overlays**. Then select **Plot Mesh** to open the **Create Mesh Plot** window.

The window is used to specify choices for the mesh plot (e.g., plot name, related solution, which nets/layers to plot).

- **Show nets selected for plot** always keeps columns for nets that are selected on any layer visible — even if they are not normally shown because of the current net filter.
- **Net filter** involves enhanced wild card filtering and is used to select which nets are shown in the window. The two wild cards (i.e., characters) supported are ***** and **vcfg**. Refer to **Regular Expressions for Nets** in **"Importing ODB++ in Ansys Electronics Desktop" on page 5-85**.

There is a shortcut menu in the window that gives options for selecting the nets for various layers. There is also a menu item for changing the column size so that net names are fully visible. After creating a mesh plot, a new Mesh item appears under the **Project Tree**.

The shortcut menu for the Mesh item is the parent to all the existing mesh plots.

- **Delete** – Deletes all the mesh plot items.
- **Plot Mesh** – Opens the Create Mesh Plot window.
- **Modify Attributes** – Open the Modify Attributes window.
- **Update Plots** – Refreshes the mesh if it is invalid.

The shortcut menu on a mesh plot **Project Tree** item:

- **Plot Visibility** – Turns on/off the mesh plot drawing in the layout.
- **Plot Mesh** – Opens the Create Mesh Plot window.
- **Modify Attributes** – Open the Modify Attributes window.

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 Mesh Seeding

Each HFSS Setup can be assigned a sequence of mesh operations. These operations share many of the same HFSS meshing functionality and features. HFSS 3D Layout supports both length-based mesh refinement and skin depth-based mesh seeding.

Seeding is the technique of guiding HFSS through the construction of the mesh. HFSS starts with an initial mesh and by adaptive refinement refines the mesh. However, if you know before starting the simulation where such refinement is needed in the model, seeding can provide a more refined starting mesh for the adaptive refinement process, which may result in obtaining the converged mesh faster.

Due to the manner in which HFSS 3D Layout pre-processes geometry and assembles mesh entities, mesh operations are assigned to net-layer pairs. Mesh seeding and refinement is added through the **Project Manager** window by right-clicking on the project's HFSS Setup analysis:

- Adding length-based mesh seeding
- Adding skin depth-based mesh seeding

Adding Length-Based Mesh Seeding to an HFSS 3D Layout Design

Length based mesh refinement controls the length of the tetrahedral elements and the growth of the whole mesh. Set a ceiling on the mesh growth with the option **Restrict the Number of Elements** to ensure a mesh of reasonable size after the length refinement.

When a mesh operation is defined and a maximum length is set, on a selection of nets and layers (i.e., more than one), the operation occurs on the whole mesh. If only one layer or net is selected, when its boundaries are refined, elements are added both inside and outside of that object.

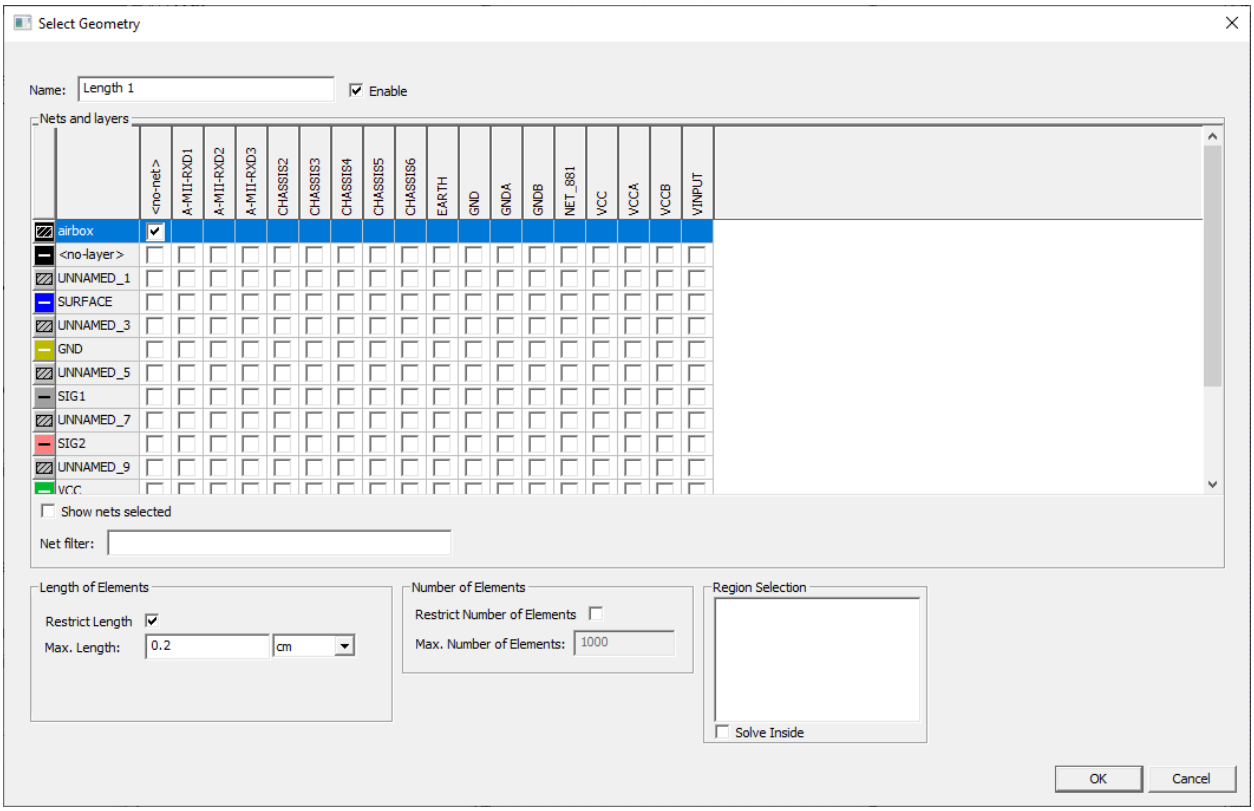
When there are no more elements to refine or when no more elements can be added, the refinement process stops because it has either met the growth limit or the element length target. However, there are more quality improvement swaps, smoothing, and matching boundary issues, performed at the finishing stage that might slightly alter the mesh size.

> **Important:**
>
> Be careful about specifying length targets for voluminous structures. This is because element count and target length have a cubic relationship. The relationship between length target and element count is inversely

1. From the **Layout Editor**, pick any selectable(s) on the net-layer pair(s) for assignment. This is optional. The selection are used to initialize the assignment 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. From the **Project Manager** window, right-click project' s HFSS Setup analysis.
3. Choose **Assign Mesh Operations** > **On/Inside Selection** > **Length Based** to open the **Select Geometry** window.



4. Select the appropriate net-layer pairs, including the airbox, and assign the seeding attributes. If necessary, display and filter for nets.
5. For **On Selection**:
    a. To restrict the length of tetrahedra edges touching the faces:
        1. Select **Restrict Length of Elements**.
        2. Type the maximum length of the tetrahedral edges touching the faces in the **Maximum Length of Elements** field. HFSS refines the element edges touching the selected faces until their lengths are equal to or less than this value.
    b. To restrict the number of elements added during refinement of faces:
        1. Select **Restrict the Number of Elements**.
        2. Enter the **Maximum Number of Elements** to add. When the mesh is generated, the refinement criteria specified is used. If the maximum number of elements is reached, some elements may exceed the requested maximum element length.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. For **Inside Selection**:
    a. To restrict the length of the tetrahedral element edges inside the object:
        1. Select **Restrict Length of Elements**.
        2. Type the maximum length of the edges inside the selection in **Maximum Length of Elements** field.
    b. To restrict the number of elements added during the refinement inside the object:
        1. Select **Restrict the Number of Elements**.
        2. Enter the **Maximum Number of Elements** to add.
7. If seeding the mesh by region, use the **Region Selection** area to select one region.
8. Click **OK**.

Adding Skin-Depth Mesh Seeding to an HFSS 3D Layout Design

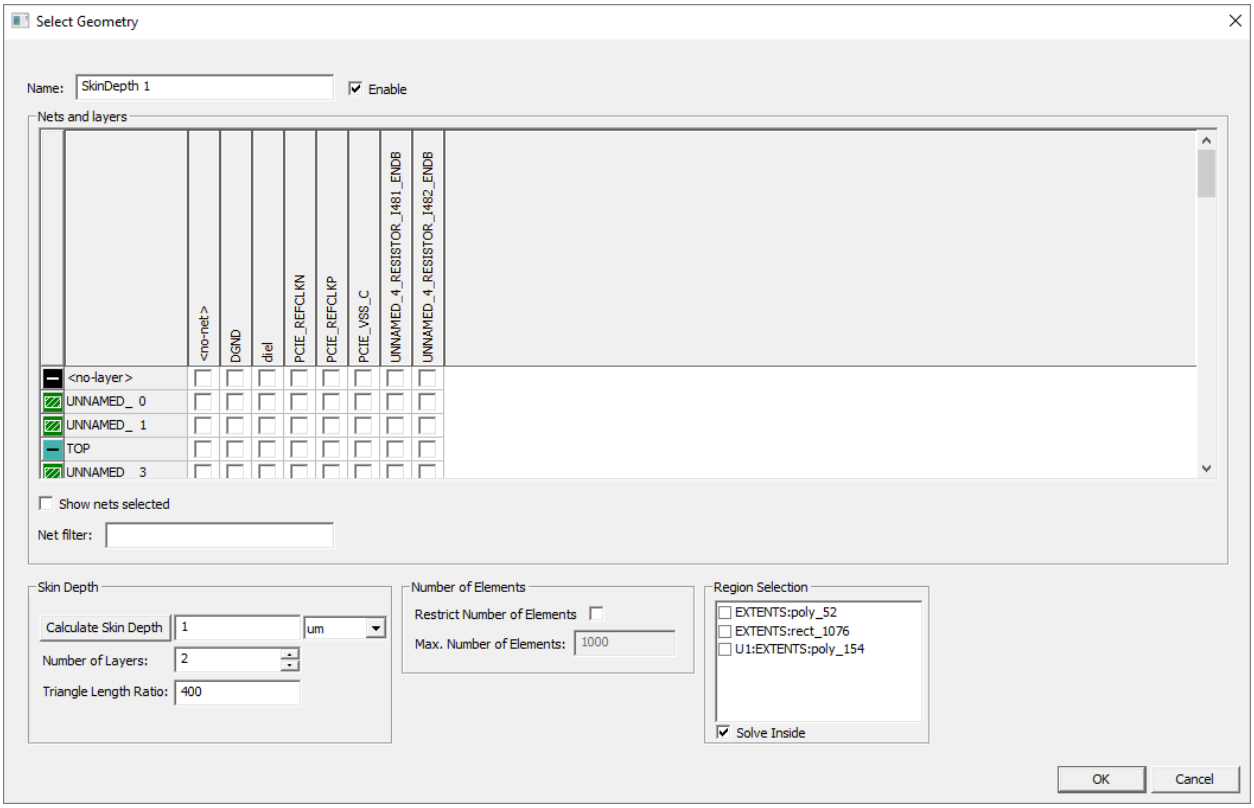
When the mesh is generated, the specified refinement criteria is used. This operation are approximately the same as having slabs of tetrahedra, but it is not guaranteed to prevent tetrahedra from crossing slab interfaces. Use caution with this mesh operation, as very thin layers may cause a reduction in mesh quality or unnecessarily cause the generation of a very large mesh. Also be careful because further regions refined under this operation and its close neighbors do not participate in solution adaptive refinement.

1. From the **Layout Editor**, select faces to refine.

    > **Note:**
    >
    > Select a body, convert it to selecting all faces of the body, and toggle a few faces to remove them from selection. Selecting the whole body might select very large regions for refinement and increase the element count a lot.

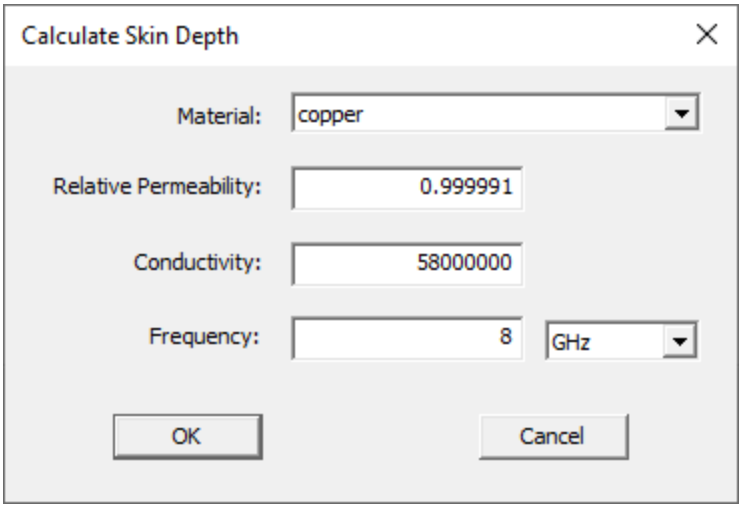
2. From the **Project Manager** window, expand the **Project Tree** and **Analysis** folder. Then right-click the HFSS Setup analysis and select **Assign Mesh Operation** > **On Selection** > **Skin Depth Based** to open the **Select Geometry** window.

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 appropriate net-layer pairs and assign the seeding attributes. Then display and filter for nets.
4. To calculate the skin depth based on the object's material permeability and conductivity and the frequency at which the mesh are refined, click **Calculate Skin Depth**. The **Calculate Skin Depth** window opens with values based on the selected object's material properties and the solution setup frequency.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



a. Select the appropriate **Material:** from the drop-down menu.
b. If selected, edit the material's **Relative Permeability:**, **Conductivity:**, or **Frequency:** by entering new values in the appropriate fields.
c. Click **OK**. The solver calculates the skin depth and enters its value in the **Skin Depth** field. Accept the calculated values or provide others.

5. In **Number of Layers of Elements**, enter the number of layers to add perpendicular to the object's surface. The skin depth is the total depth of all layers combined. The solver adds an equivalent number of mesh points to each layer (e.g., if HFSS added 10 points to satisfy the **Triangle Length Ratio**, it adds 10 points to each layer).
6. Optionally, provide the length ratio of the surface mesh in **Triangle Length Ratio**. The default value is set to 400. The solver refines the surface triangle mesh (the faces of the tetrahedra touching the surface) until their edge lengths are less than or equal to a value determined by the formula: **Triangle Length Ratio** * **Skin Depth** / **Number of Layers of Elements**.
7. To restrict the number of surface elements added during refinement of faces:
   a. Select **Restrict the Number of Surface Elements**.
   b. Enter the **Maximum Number of Surface Elements** to add. When the mesh is generated, the refinement criteria specified is used. If the maximum number of elements is reached, some elements may exceed the requested maximum element length ratio.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

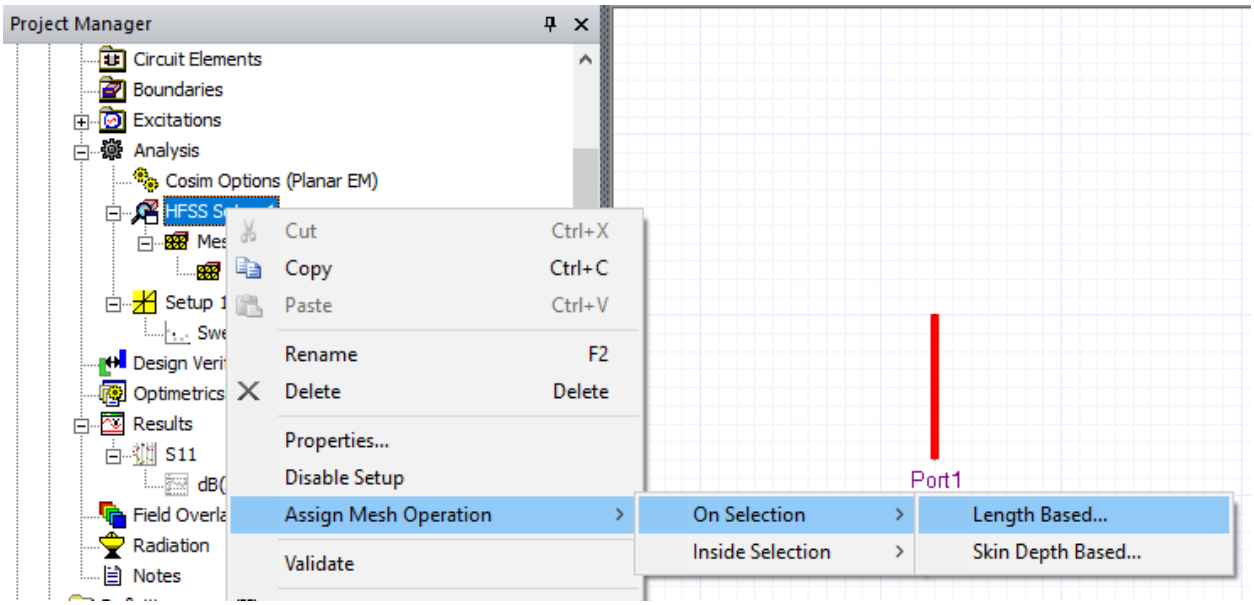
> **Note:**
>
> By default, **Restrict Number of Surface Elements** is unchecked to allow the mesher to use symmetry more effectively. Restricting the number of surface elements may affect symmetry but can protect against runaway refinement in specific cases. When **Restrict Number of Surface Elements** is unchecked, the **Maximum Number of Surface Elements** is unavailable and provides an estimate of mesh growth.

8. If seeding the mesh by region, use the **Region Selection** area to select one region.
9. Click **OK**.

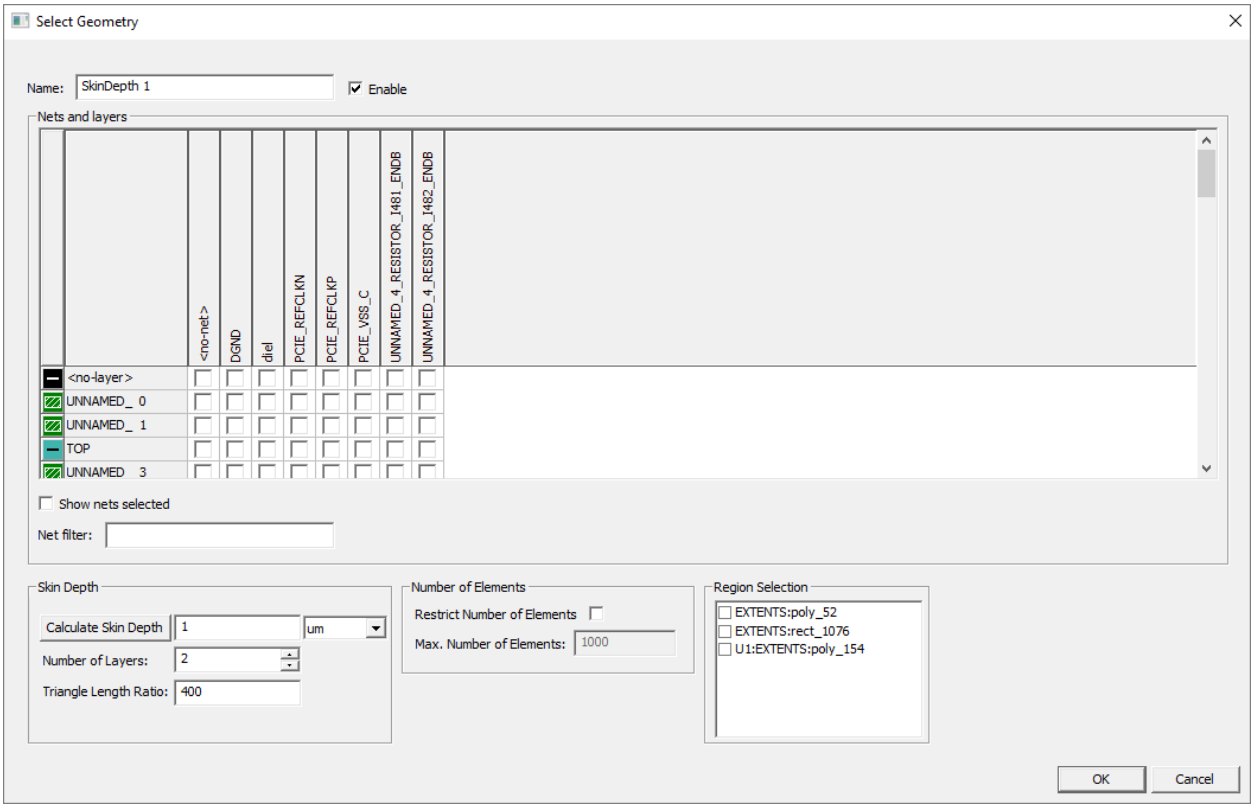
The procedure is complete.

Seed Refinement for Mesh Growth

Seed refinement for mesh growth is performed by defining either the **Number of Elements** and/or the **Length of Elements** option. When either one of these specified criteria on the **Select Geometry** window is met, seed refinement stops. To access the settings for seed refinement, in the Project Manager choose **Assign Mesh Operations** > **On Selection**. Depending upon your selection, you can either seed inside the object or on the object surface.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

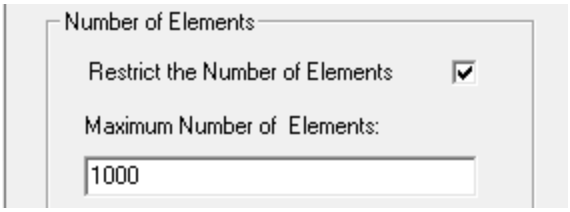


Mesh Refinement Scenarios

This section describes how mesh refinement settings affect your simulation. You can refine a mesh at any stage and then solve from there.

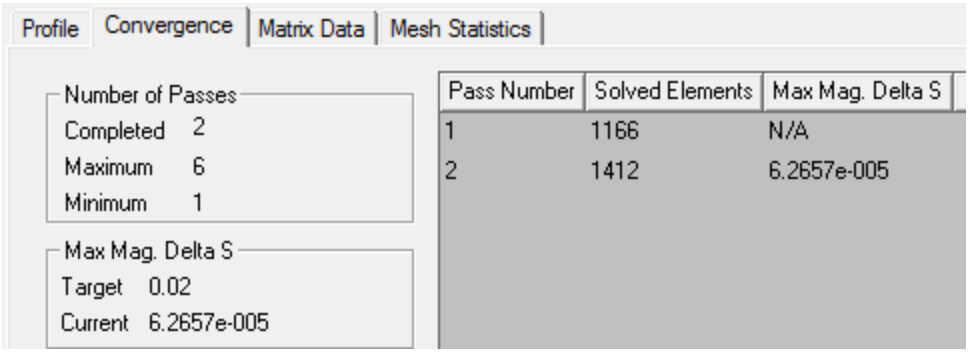
Assign Mesh Operations Upfront

In this scenario for a cube, the mesh operation is performed upfront.

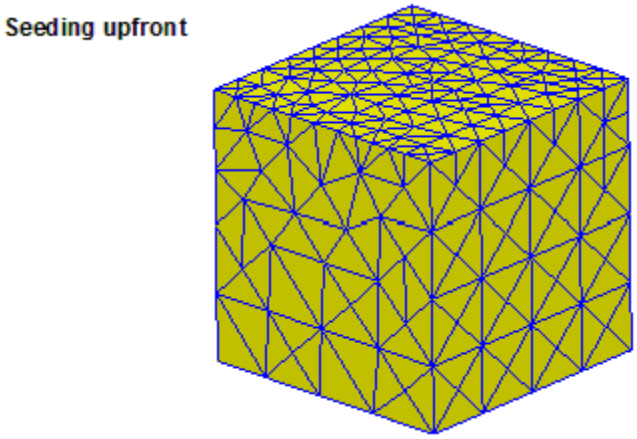


Therefore the assigned mesh operation is applicable for the first pass and 1000 elements are added in the initial mesh as shown in the following figure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The mesh plot for this operation is shown in the following figure.



Note:

The number of solved elements is directly affected only for that pass where seed operation begins. For example, when seeding is performed upfront, only the first pass gets affected and elements are added according to the value defined in the mesh for the first pass only, and subsequent passes are solved but they remain unaffected directly by the seed operation. Similarly, if you seed from the nth pass, the operation affects only the nth pass and the subsequent passes from n+1 are solved from there without being affected directly by the seed operation. For example, if you set **Maximum Number of Elements** = M for a particular pass N, M elements are added on the Nth pass only. In the (N+1)th pass, again M extra elements are not added.

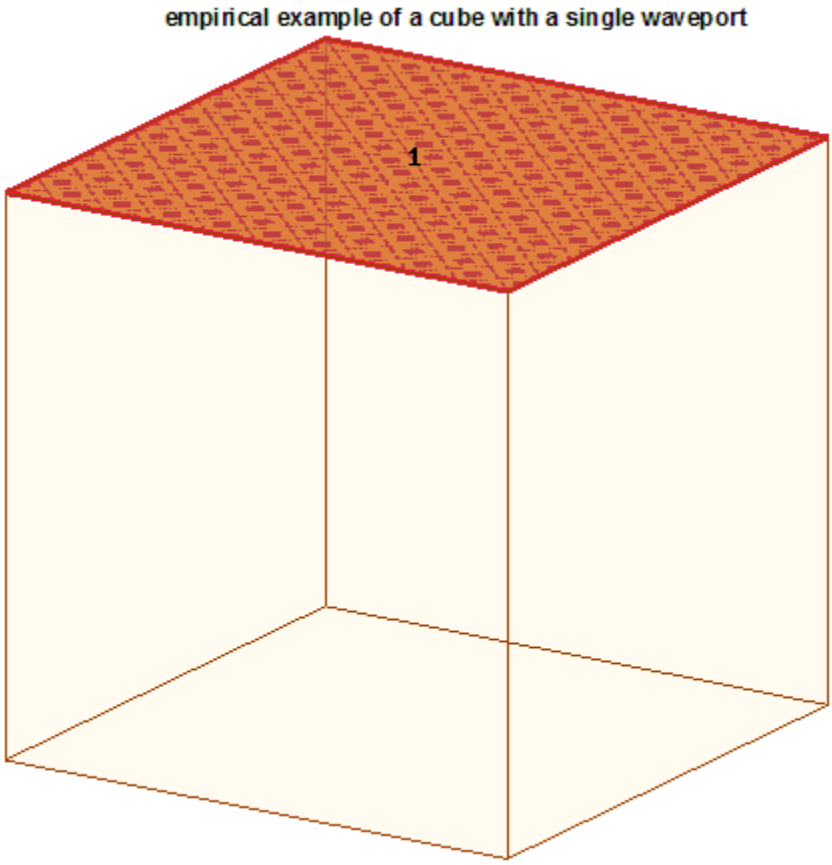
Restrict Length of Elements

Selecting the **Restrict Length of Elements** check box enables you to set the stopping criteria for seed refinement. When the longest edge of a tetrahedral element under the specified value

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

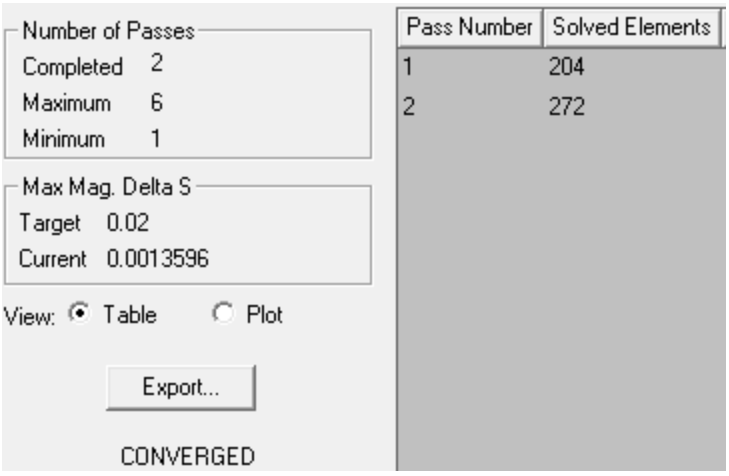
of **Maximum Length of the Tetrahedral Element** is reached, seed refinement stops.

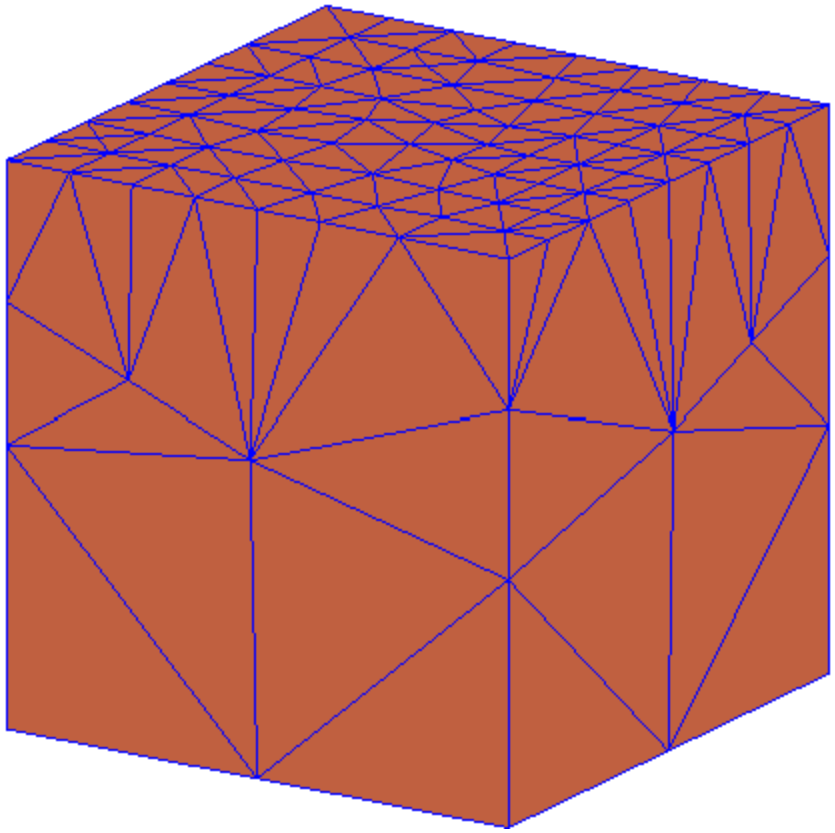
The following example of a cube 1000 mm in size and material vacuum illustrates seed refinement when it is specified by a value of the **Maximum Length of The Tetrahedral Element** as the stopping criteria. The cube has a wave port assigned on one of its faces and the adaptive frequency is 0.25 GHz.



First the design is solved with no seed refinement. The mesh plot on the cube after convergence is shown in the following figures.

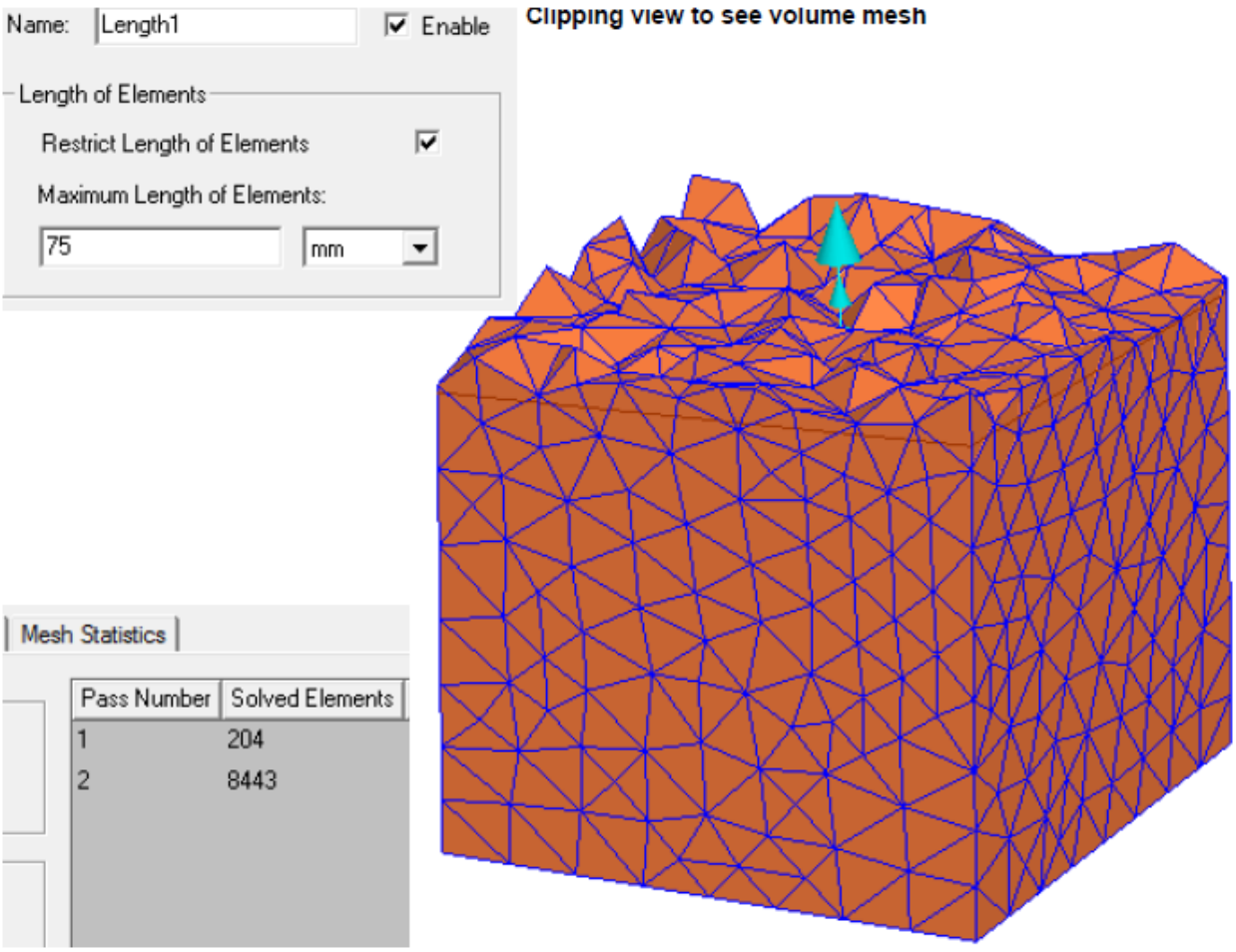
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





A plane is drawn at the center of the cube and the normal to the plane is along the Z axis. Using the **Clipping** view feature, the cube is virtually clipped along the normal. After seed refinement, the mesh plot of the cube along the normal is 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.

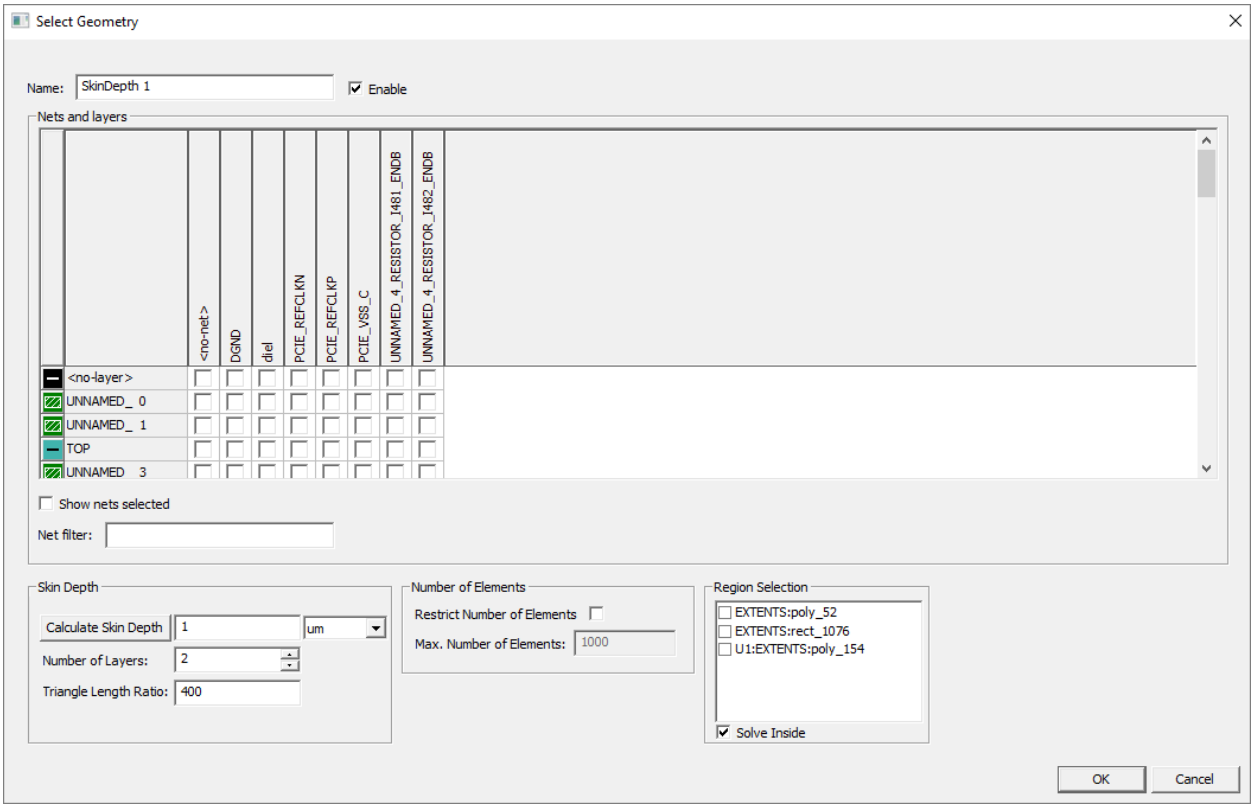


Seed Mesh by Region

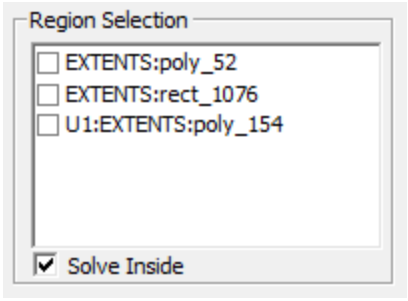
To constrain a mesh seeding operation to a geometric region:

1. In the design, draw a rectangle, circle, or polygon on a user layer.
2. Add a solution setup to the design.
3. From the **Project Manager**, right-click the solution setup and select **Assign Mesh Operation**.
4. Select any of the **On Selection** or **Inside Selection** options to open the **Select Geometry** 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. Use the **Region Selection** area to select the appropriate geometry in the design. Check the **Solve Inside** box to solve inside the mesh region.

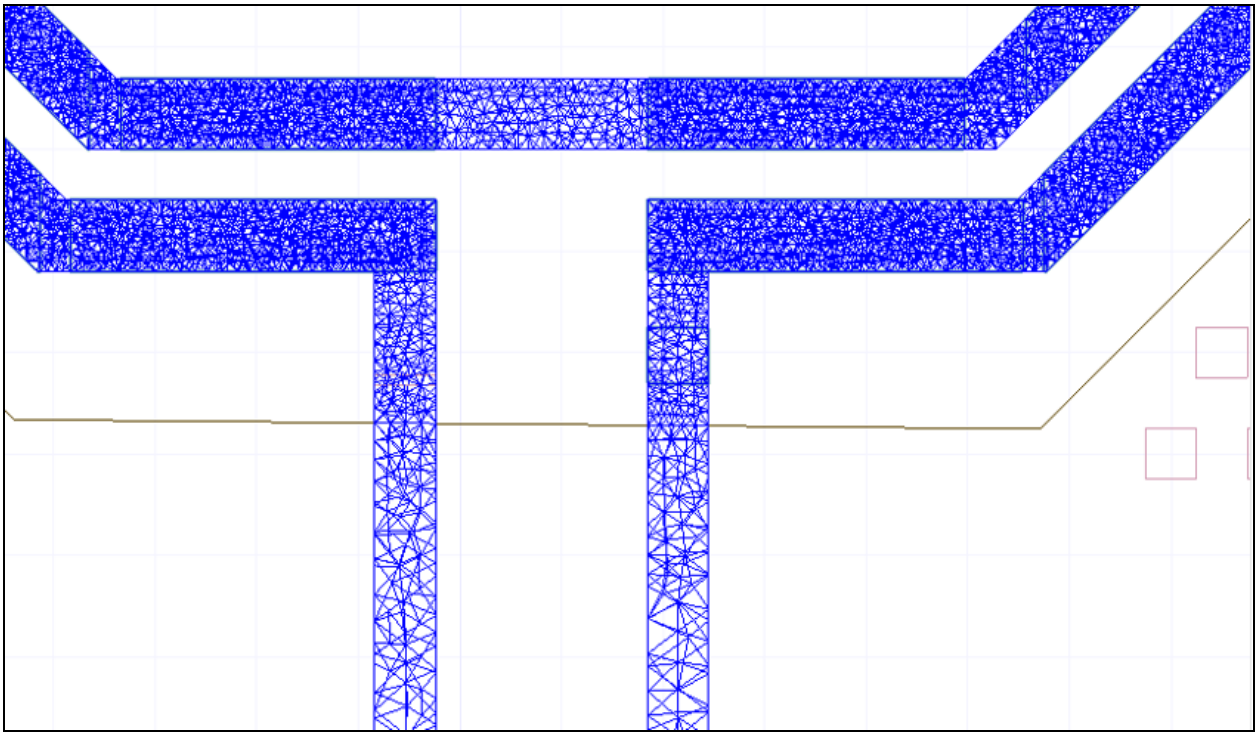


> **Note:** Select **one** region per mesh operation.

6. Click **OK**.
7. Run the simulation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Create a mesh plot. The effect of the seeding operation can be seen in the mesh plot.



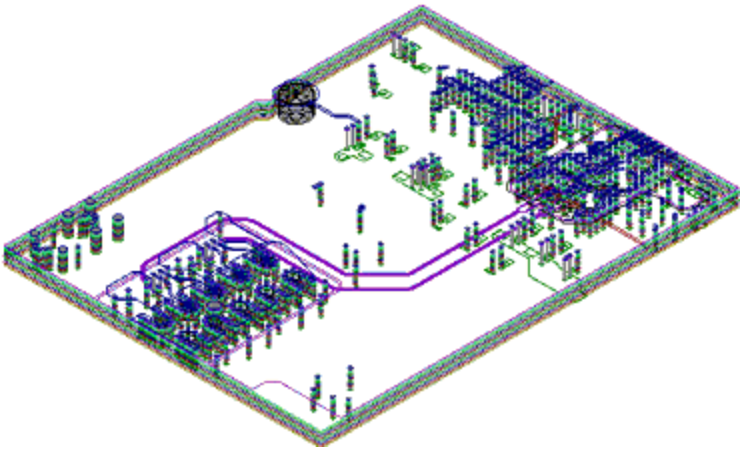
The procedure is complete.

The Phi Mesher

The Phi mesher is a layout-based meshing technology available in the HFSS 3D Layout interface. This advanced meshing technology is capable of rapidly generating an initial mesh, ensuring faster simulations that can be further accelerated and enhanced by using high-performance computing. While the traditional Classic mesher is suitable for HFSS 3D MCAD and for certain geometries solved in HFSS 3D Layout, the Phi mesher works on the HFSS 3D Layout interface where principally 2.5D geometry is comprised of z-extruded layers that have no oblique facets. Therefore, package models containing bondwires cannot take advantage of this meshing technology.

The Phi mesher uses a unique method to generate the initial mesh from the a priori knowledge of the layered nature of the geometry. It generates a volume mesh directly from the layout data, saving a tremendous amount of time, requiring less user cleanup, and eliminating several situations that might cause potential kernel errors in the traditional 3D modeler flow. When Phi is selected as the initial mesher, you can also specify a **Maximum delta-Z ratio** as a value or variable. This setting controls the allowable ratio between adjacent layer thicknesses in the z-direction to prevent excessively stretched elements during the extrusion process.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

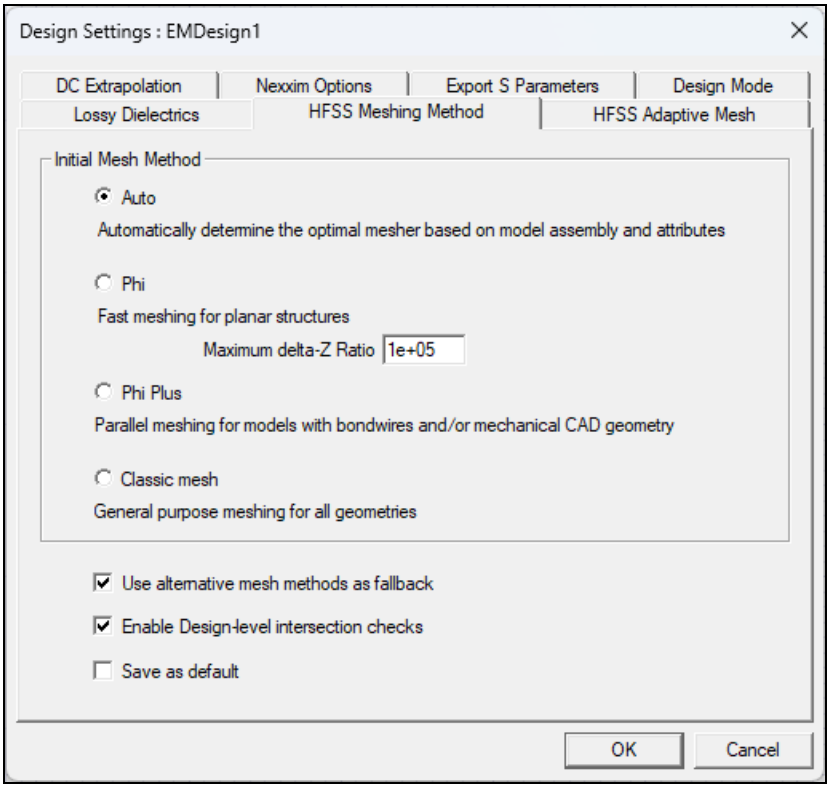


When compared to the traditional Classic mesh, the Phi mesher is several orders of magnitude faster, including from the initial meshes the adaptive meshing algorithm produces to the final convergence. Another advantage of the Phi mesher is its ability to mesh larger and more complex designs previously unmeshable from the traditional 3D interface or with the Classic mesher in the 3D Layout interface. In cases where Phi produces large meshes, you can use the HPC solver feature to leverage more cores and distributed memory to neutralize the time that would have otherwise been required for the simulation to complete.

Meshing With the Maximum Delta-Z Ratio

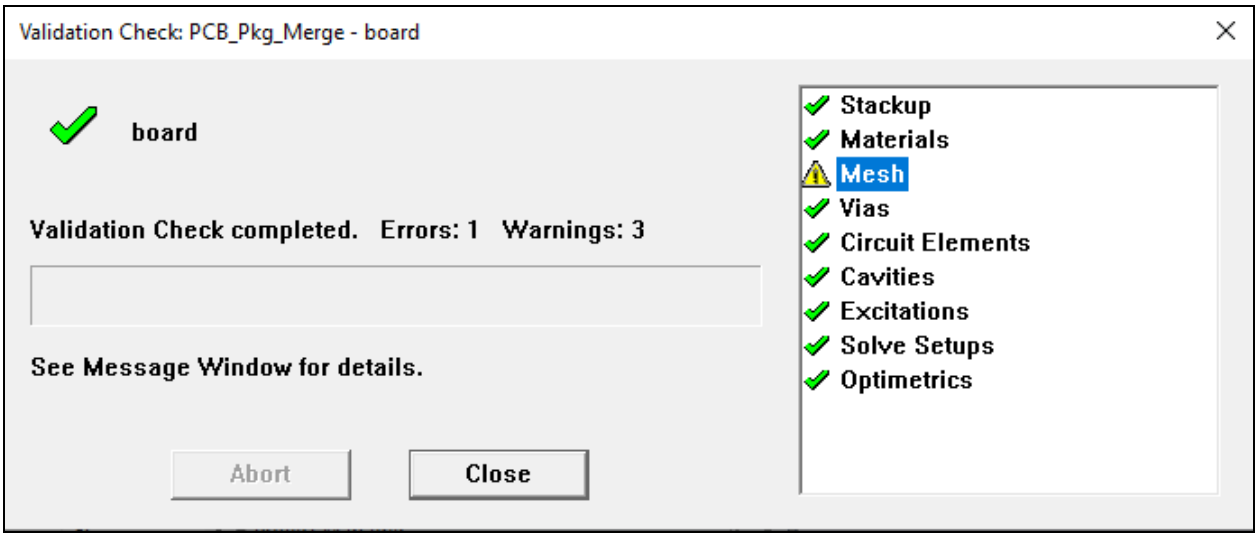
The **Phi mesher** analyzes all combinations of Z-elevation to determine the delta-Z ratio (i.e., the minimum distance between the model's unique Z-elevations). If the ratio is greater than the maximum delta-Z ratio, the validation check fails, enabling users to identify layers that are, for example, typically too thin before running a full analysis. If users deliberately make design choices that would otherwise not pass validation, entering a different value in the **Maximum delta-Z Ratio** field can allow the validation check to proceed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



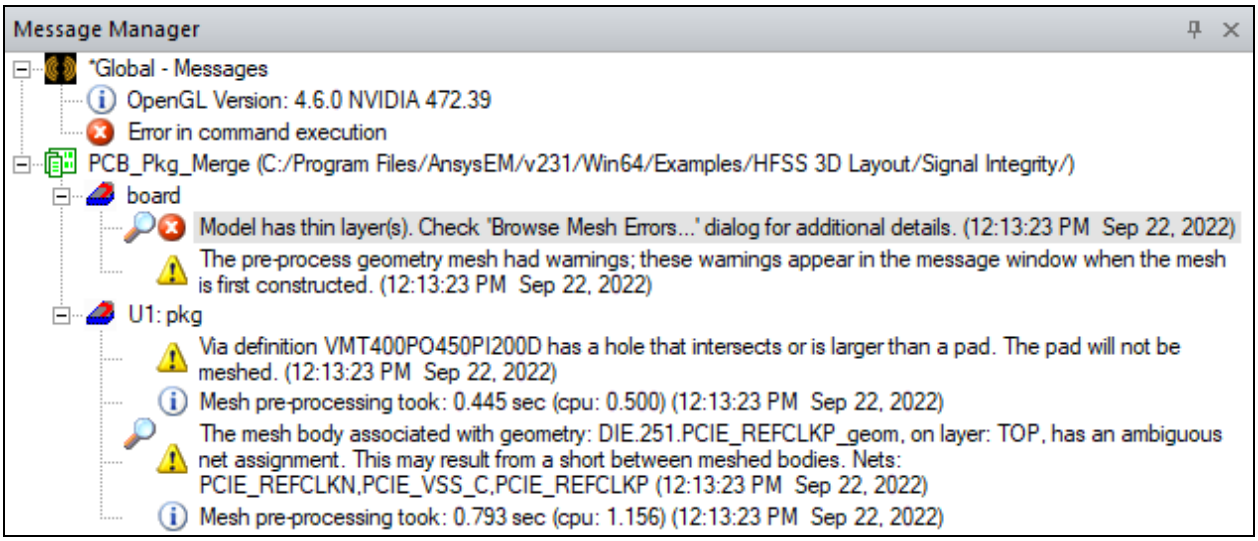
The Mesh Feedback Window

If validation fails or users wish to view the calculations performed during validation, follow these steps.

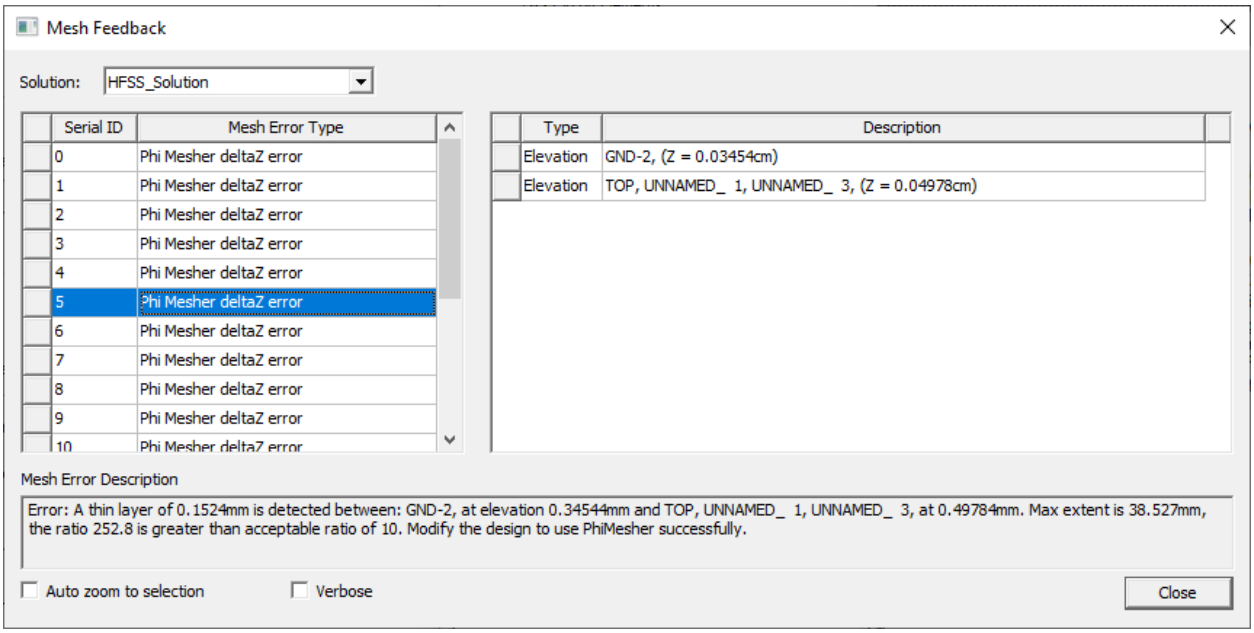


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Navigate to the **Message Manager** window and locate the message indicating the validation check's final outcome (e.g., Model has thin layers(s)).

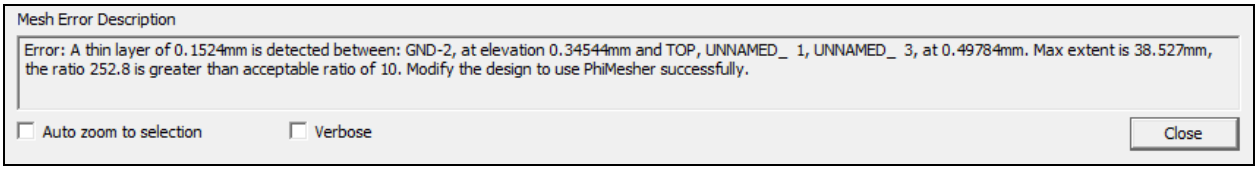


2. Double-click the validation message to open the **Mesh Feedback** window.



3. From the **Mesh Feedback** window, select from the errors listed in the left column to view its related value on the right and a description of the error in the field below.

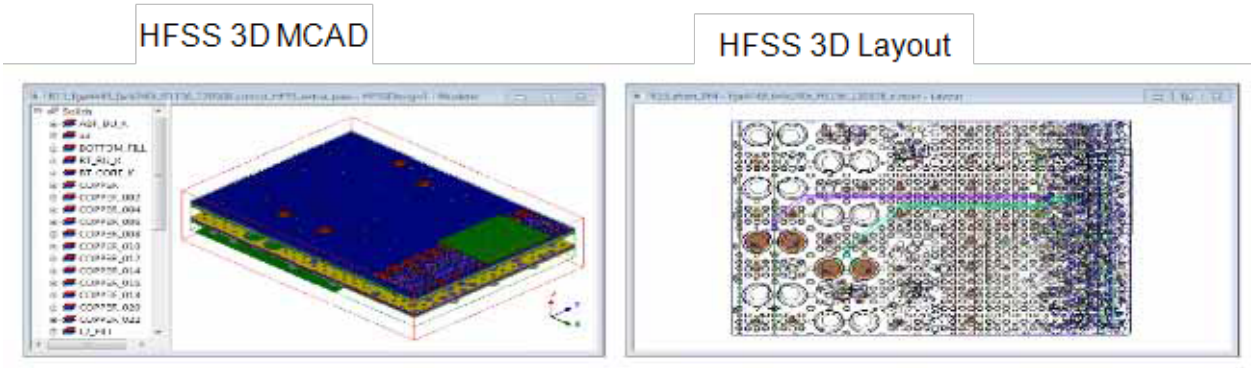
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Close** to close the **Mesh Feedback** window.

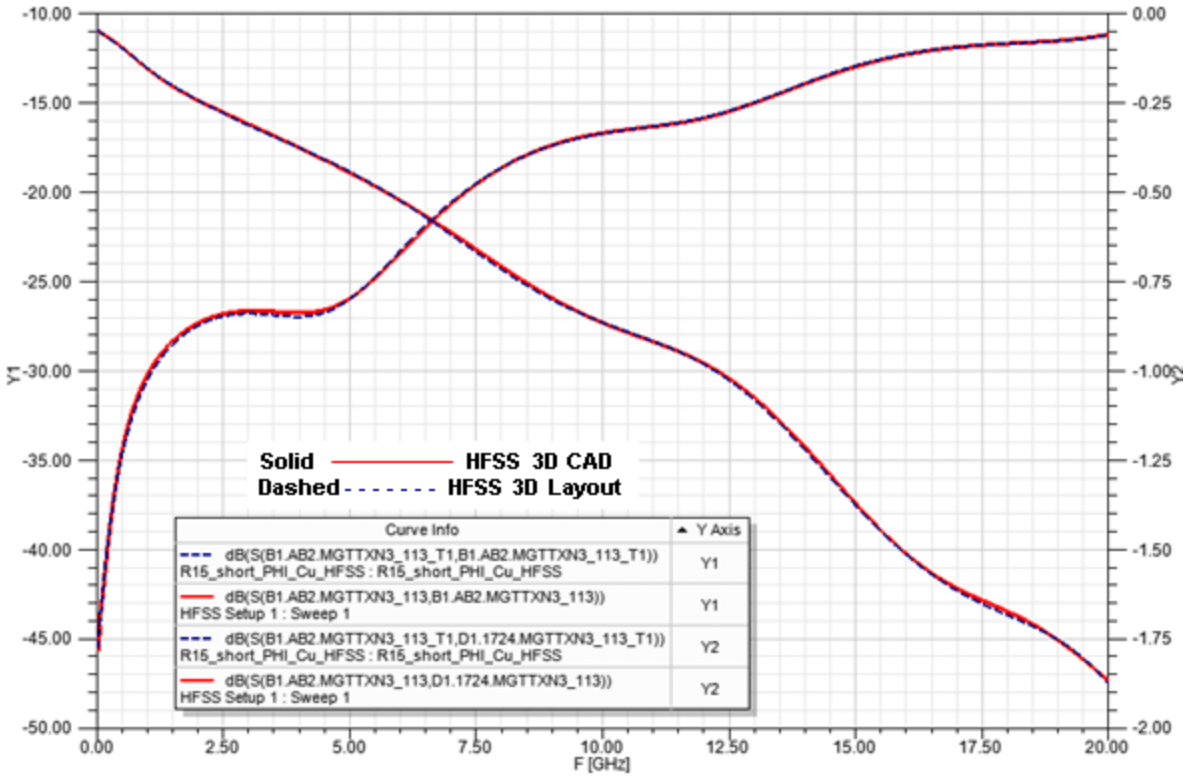
Phi Mesher for Multilayer Complex Package Layout

In this section, the simulation of a multilayer complex package layout is used to demonstrate the feature of the Phi mesher in HFSS 3D layout. To illustrate the advantages of a Phi mesher, a task-based comparison is performed between a regular HFSS 3D MCAD and the same geometry which is imported, set up, and solved through the HFSS 3D Layout GUI.



The same HFSS solver engine is used to solve both the geometries.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The HFSS MCAD results marked in red agree with the HFSS 3D Layout results in dashed blue.

The comparison of the two solutions is shown in the following table.

	HFSS 3D CAD	HFSS 3D Layout
Initial Mesher	Classic	Phi
Initial Mesh Size (tets)	763, 953	712, 536
Initial Mesh Time	1hr 20min	4min
Number of Adaptive Passes	6	7
Number of Tets - Final	1, 617, 954	1, 620, 969
Adaptive Passes Run Time	4hrs 1min	1hr 5min
RAM	47.6 GB	49.6 GB
HFSS Model number of Ports	8	8

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

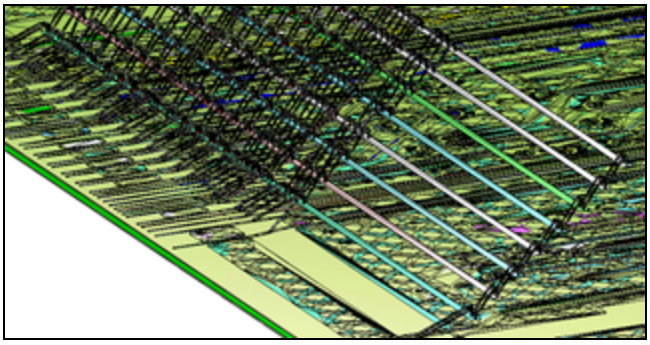
Initial mesher in the HFSS 3D CAD interface takes 1 hour and 20 minutes to generate the initial mesh. The entire simulation takes just over 4 hours to complete in 6 adaptive passes, and the total number of tetrahedra generated in the final mesh is 1.6 million.

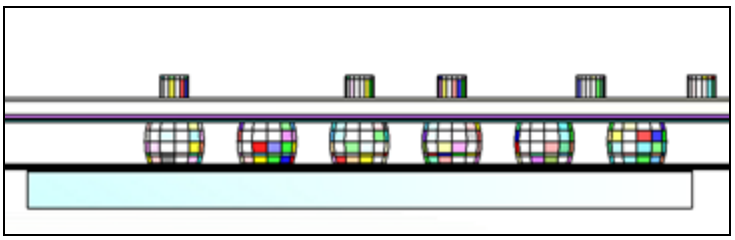
On the other hand, the Phi initial mesher creates a very comparable initial mesh size of 712000 tetrahedra in only 4 minutes. The entire simulation takes one extra pass to solve; however, for the same number of solved tetrahedra, 1.6 million, it required only 1 hour and 5 minutes. This example shows the benefits of using the HFSS 3D layout and its ability to leverage the Phi mesher to mesh an extremely complex geometry. Typically, the Phi mesher requires a small amount of time ranging from only 5 to 10 minutes to create the initial mesh. The Phi mesher provides a great advantage for solving any arbitrary package or PCB layout structures.

The Phi Plus Mesher

Phi Plus is a multi-domain parallel mesher, available in HFSS 3D Layout designs. This advanced meshing technology is capable of rapidly generating an initial mesh with acceleration from high performance computing (HPC). Complex models are decomposed into smaller subdomains and meshed in parallel using multiple cores. Therefore, to use Phi Plus, specify multiple cores in the HPC options.

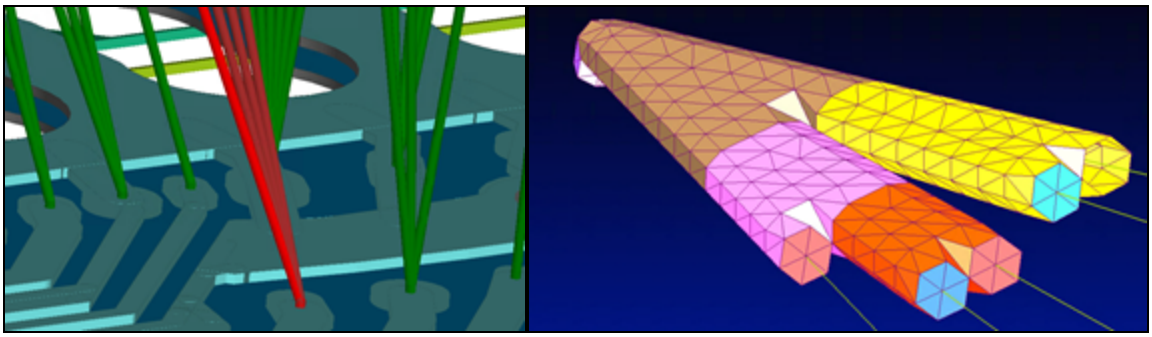
The Phi mesher in HFSS 3D Layout is a special-purpose fast mesher designed for purely layered structures. In contrast, Phi Plus works with all types of layout models, particularly those containing bondwires, solder balls, or 3D Components (MCAD). A typical complex layout package with bondwires and solder balls is shown in the following figure:



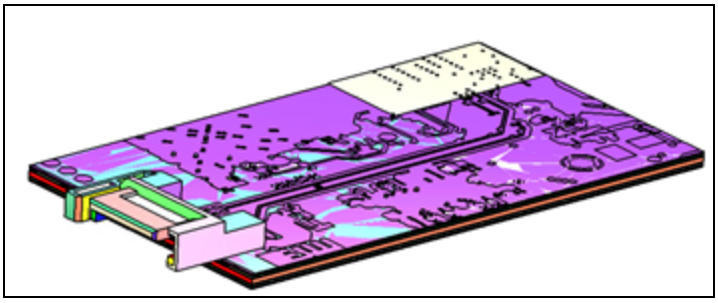


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phi Plus is particularly effective in meshing models where multiple bondwires overlap at the ends. The advanced union-simulated technology in Phi Plus creates a clean and accurate mesh for simulation:



Phi Plus efficiently creates complete meshes for assemblies of layout and 3D MCAD components:



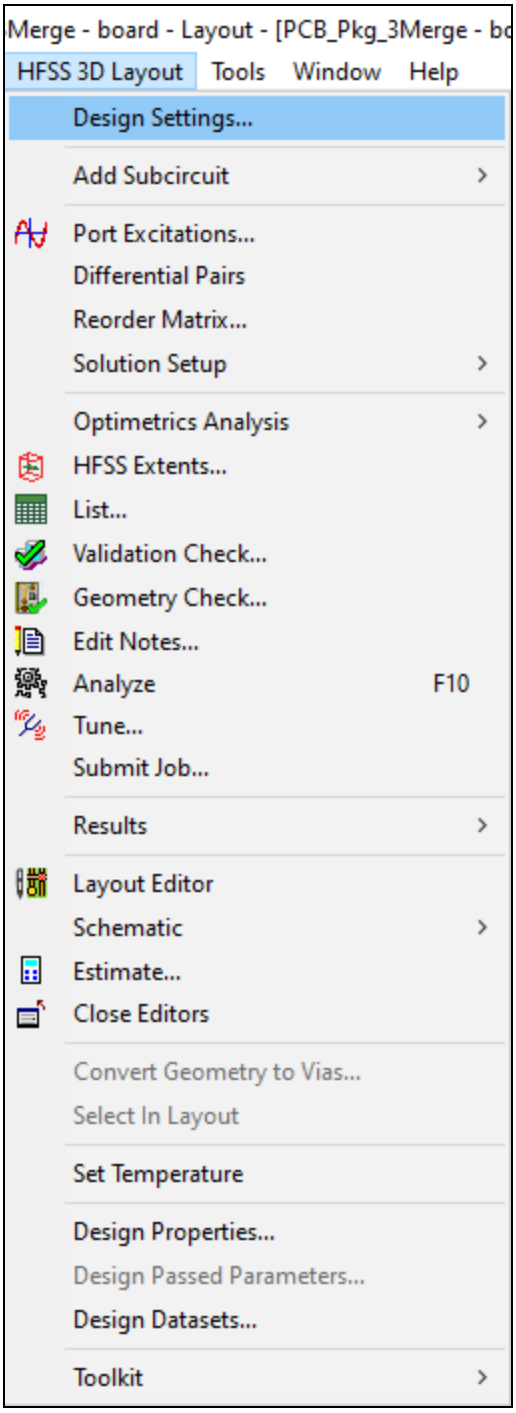
The advanced hybrid mesh technology and parallel meshing provides extremely fast, uniform, and high quality meshes that enhance solver performance. Compared to the traditional Classic mesh, Phi Plus is several orders of magnitude faster and equivalent in speed to the Phi mesher.

Selecting the Phi Plus Mesher

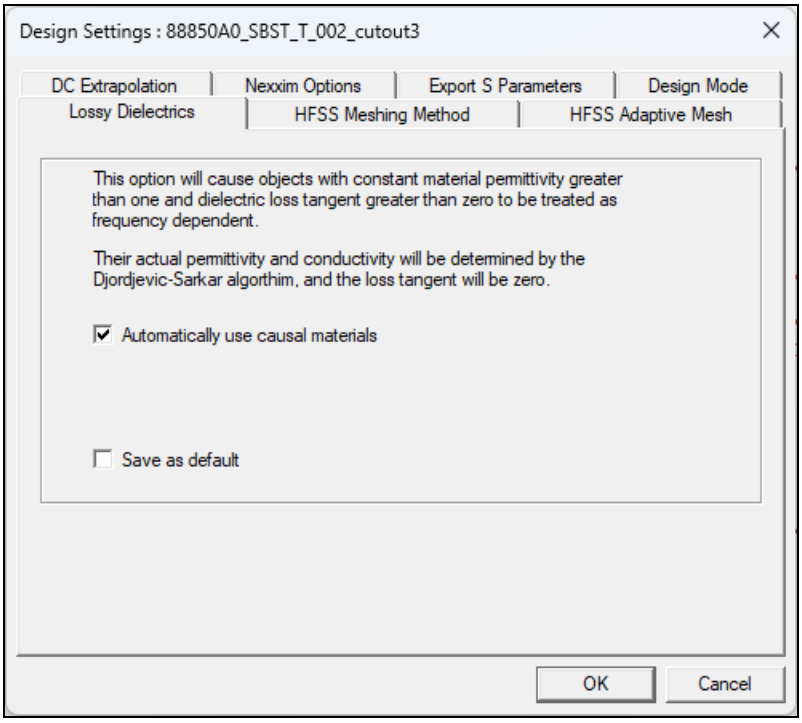
Follow these steps to select the Phi Plus.

1. From the **HFSS 3D Layout** menu, select **Design Settings** to open the **Design Settings** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

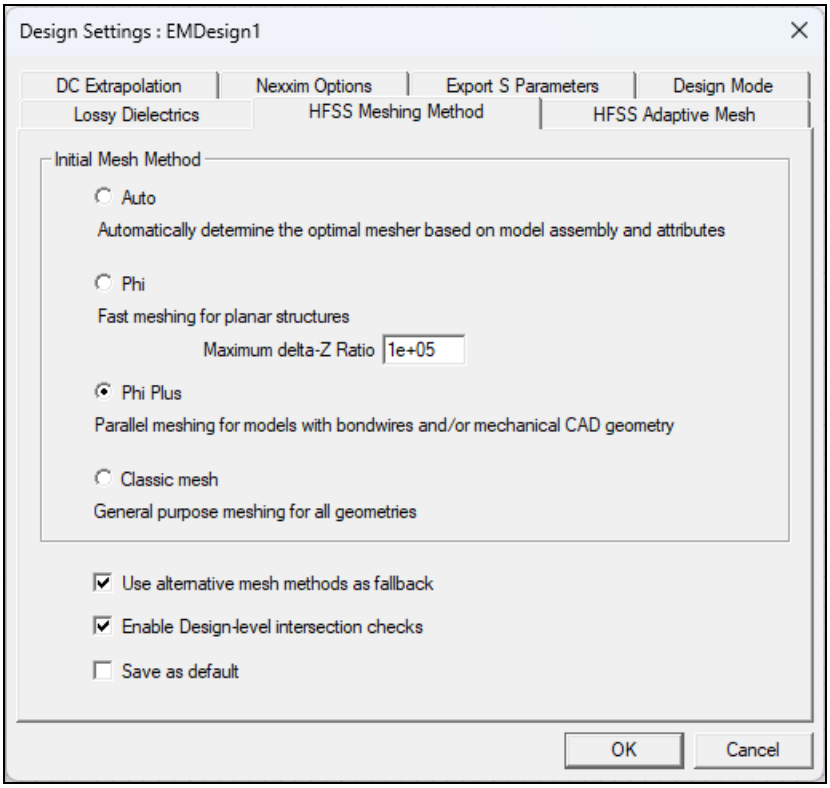


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click on the **HFSS Meshing Method** tab and choose **Phi Plus** on the **Initial Mesh Method** area.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



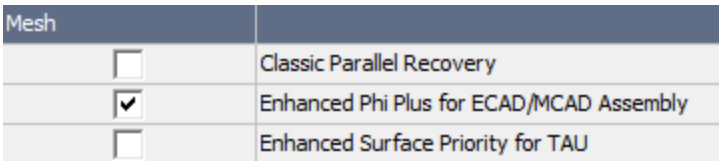
3. Click **OK** to close the **Design Settings** window.

The procedure is complete.

Enhanced Phi Plus for ECAD/MCAD Assembly

Enhanced Phi Plus is in the beta version. It streamlines the meshing algorithms to generate more reliable mesh with a more efficient workflow.

To activate it, enable Beta Option “Enhanced Phi Plus for ECAD/MCAD Assembly”.

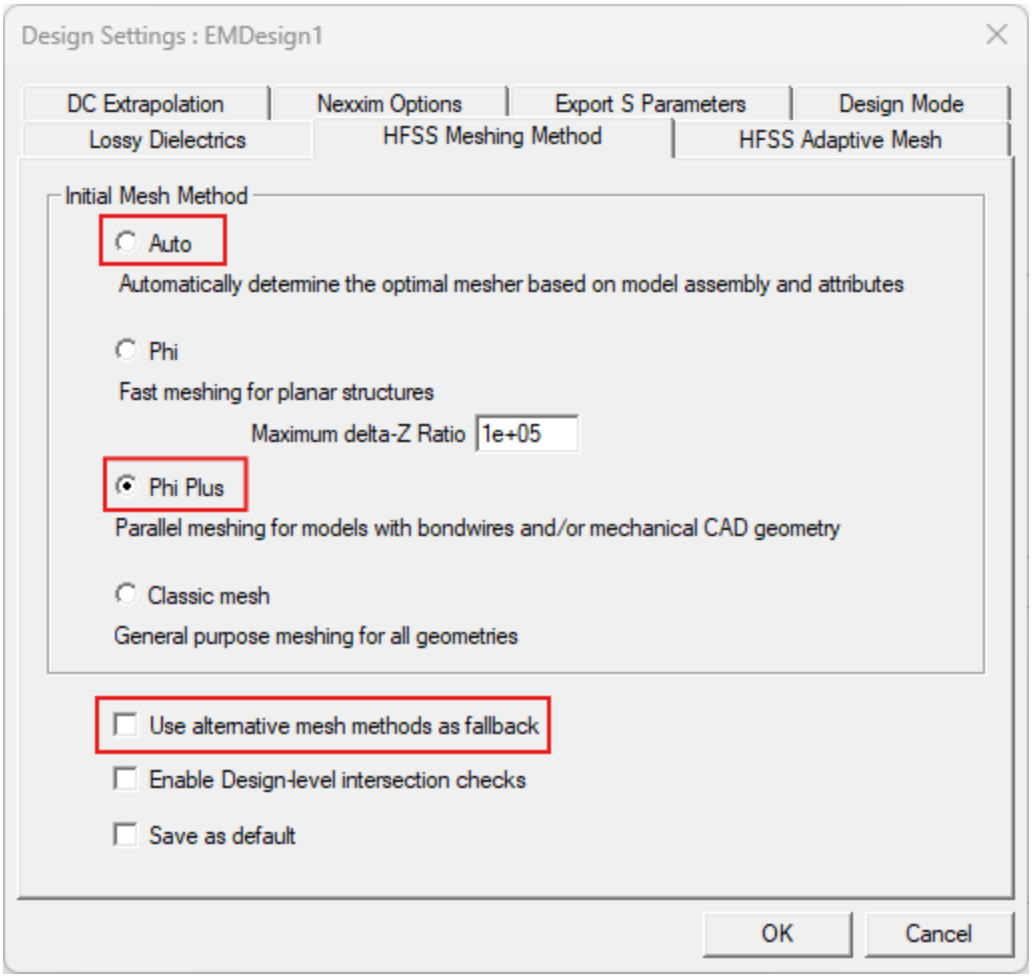


Note:

- This beta option does not support Q3D.
- This beta option only works with Phi Plus Mesher. Select “Auto” or “Phi Plus” as initial mesh method.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- This beta option does not support the fallback to alternative mesh methods. Please disable “Use alternative mesh methods as fallback”.



HFSS Mesh Simplification

A design can be solved faster by simplifying its mesh. Three options are available on the **Advanced** tab of the HFSS Setup window to reduce the number of objects in a design or its mesh:

- **Remove voids with an area smaller than** <value>: simplifies meshing by removing small voids in a design.
- **Remove Floating/Inactive Signal Net Geometry**: approximates a mesh by removing inactive geometry on the mesh.

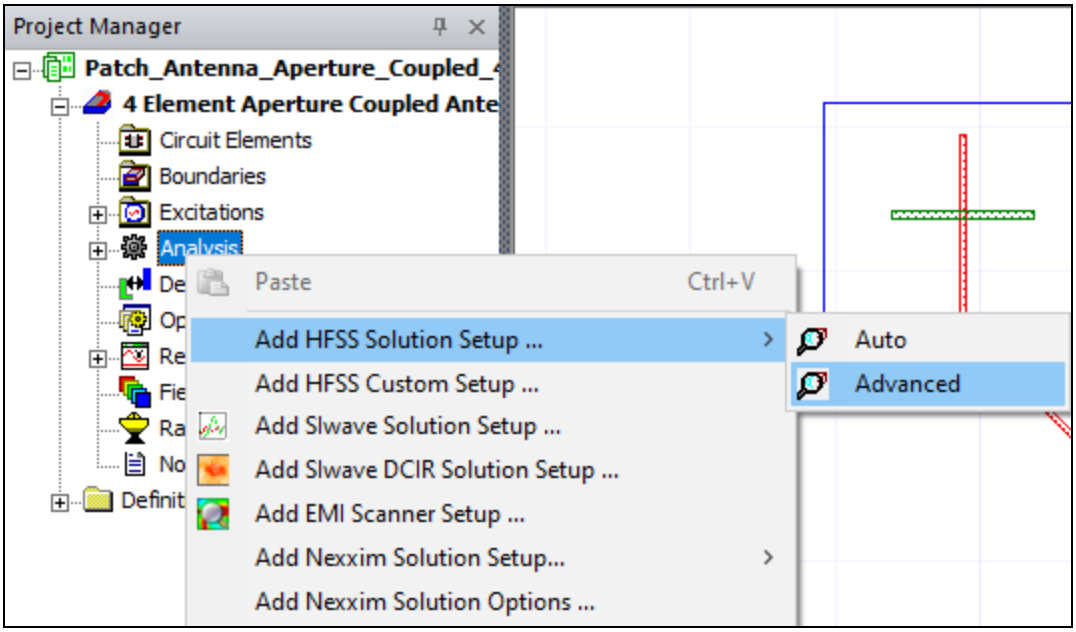
Floating and Inactive Geometry

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active geometry is geometry that plays a role in a simulation. For the purposes of the **Remove Floating/Inactive Signal Net Geometry** option, active geometry is defined as geometry that is connected to a terminal or port, or is a part of a net that is classified as a Power/Ground net. Inactive geometry is not connected to any ports nor is it part of any Power/Ground nets. Inactive geometry may be connected to the reference plane of a component. Inactive geometry is referred to as floating because it is not connected to a port or terminal. When HFSS 3D Layout filters inactive geometry out of design, it simplifies the design's mesh which causes the design to solve faster.

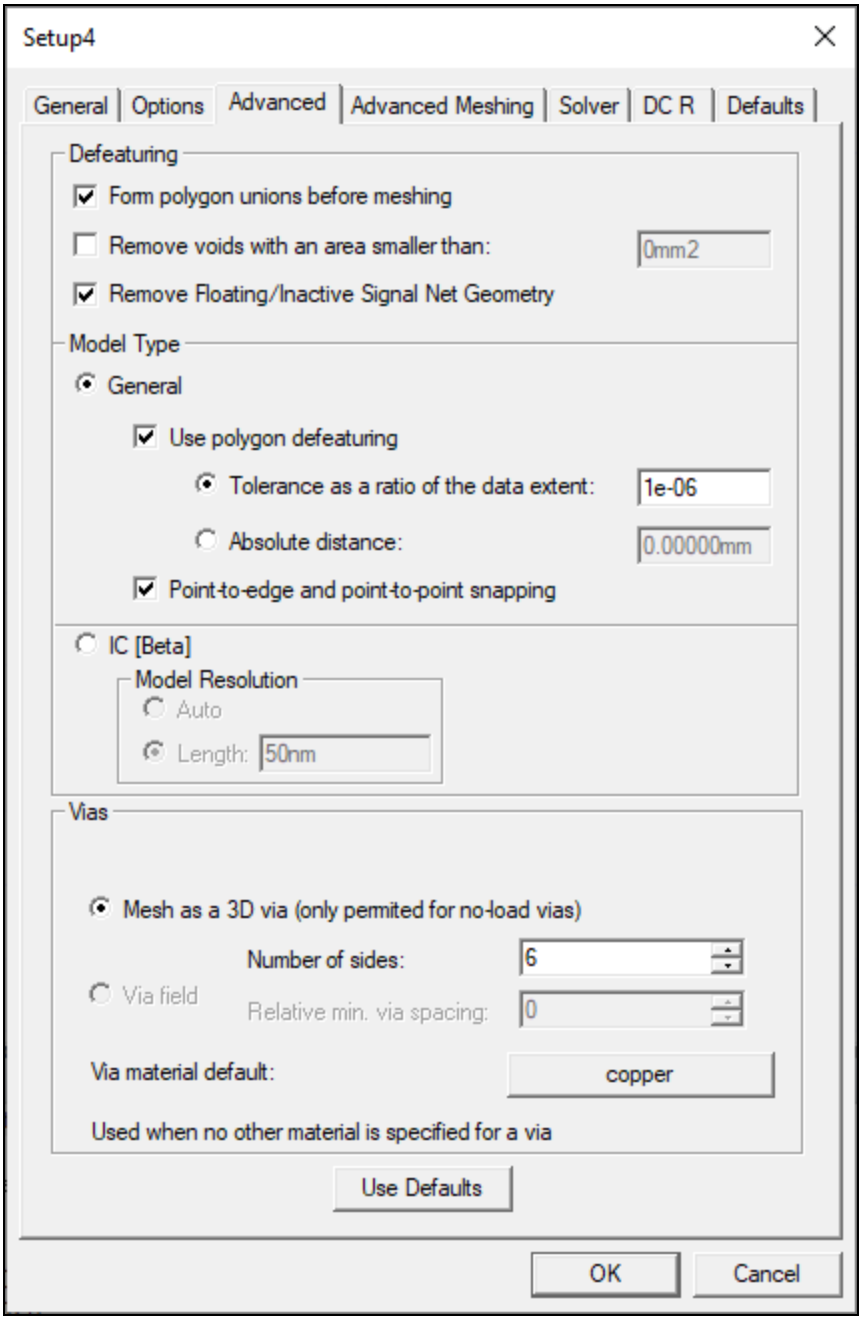
To filter out floating and inactive geometry:

1. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** and select **Add HFSS Solution Setup** > **Advanced** to open the **Setup*n*** window.



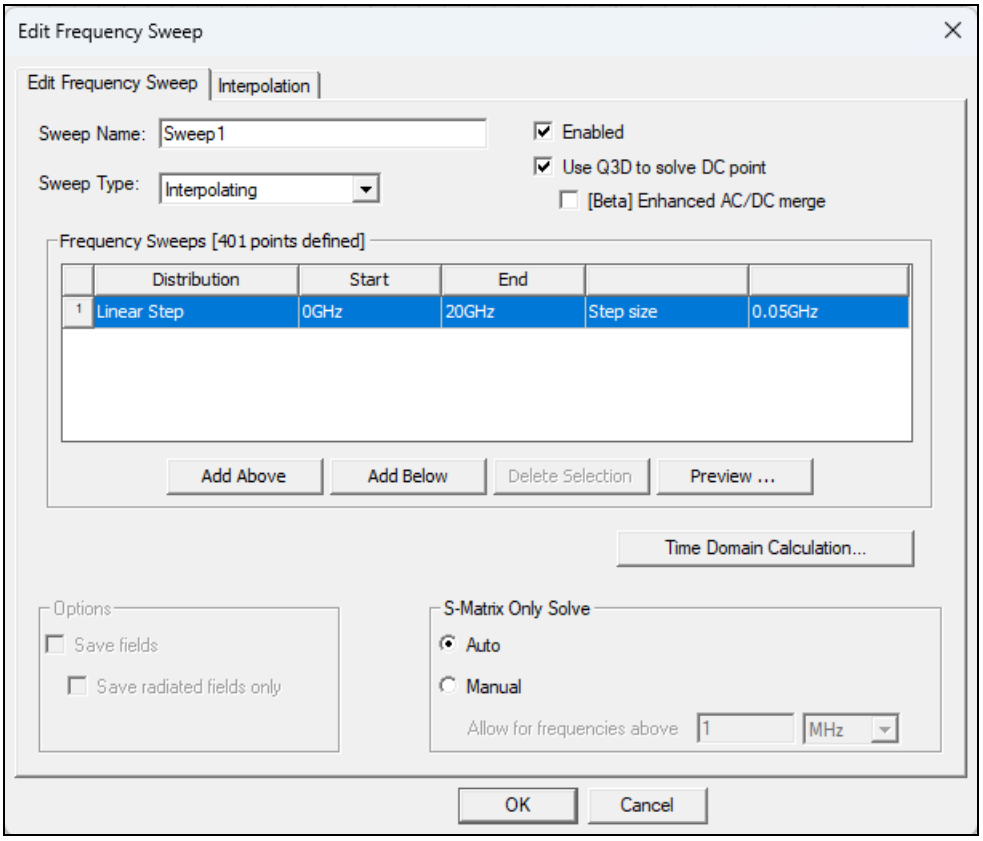
2. From the **Setup *n*** window, select the **Advanced** tab. Then click **Remove Floating/Inactive Signal Net Geometry** in the **Defeaturing** area.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



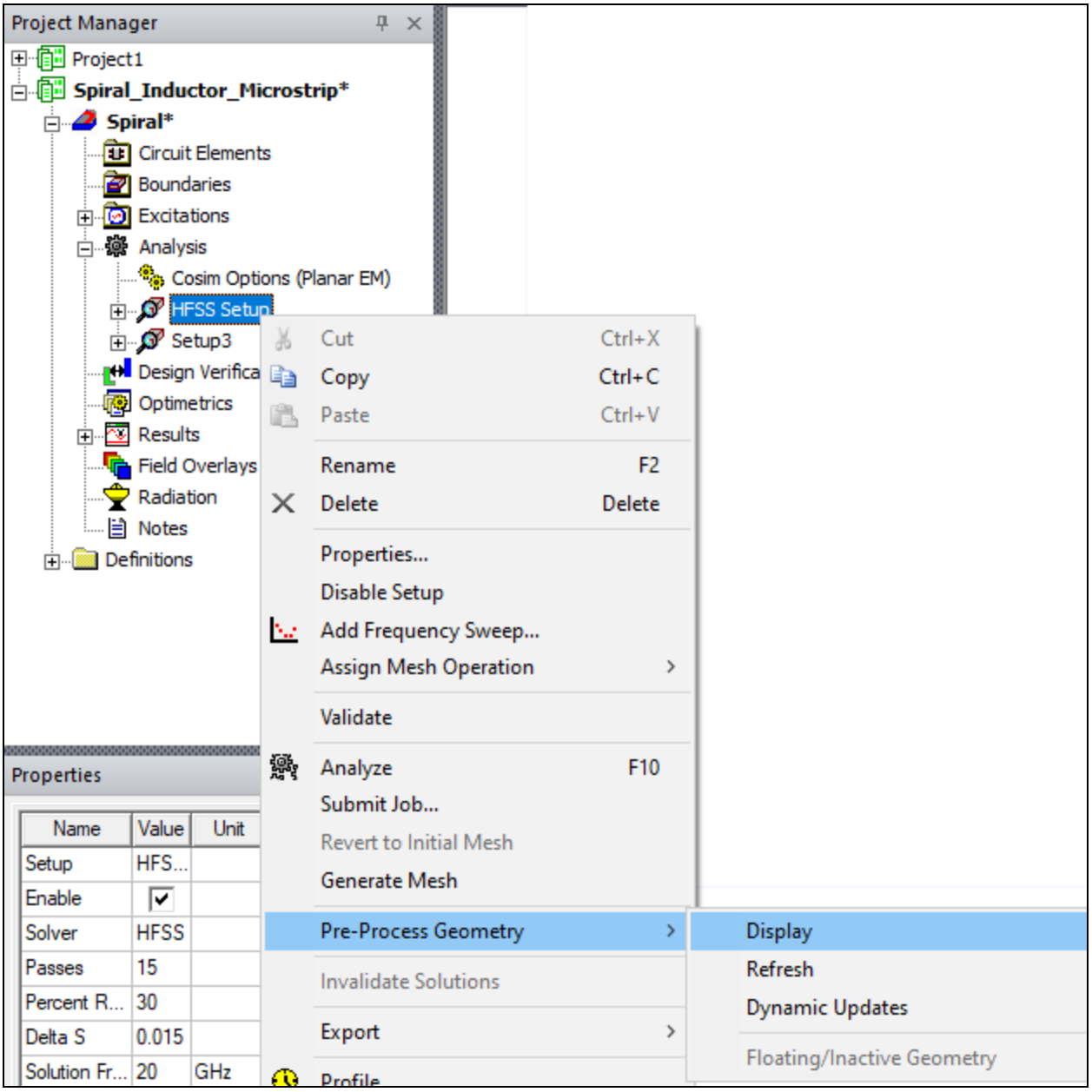
3. Click **OK** to close the **Setup*n*** window and open the **Edit Frequency Sweep** window. Click **OK** to close the window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



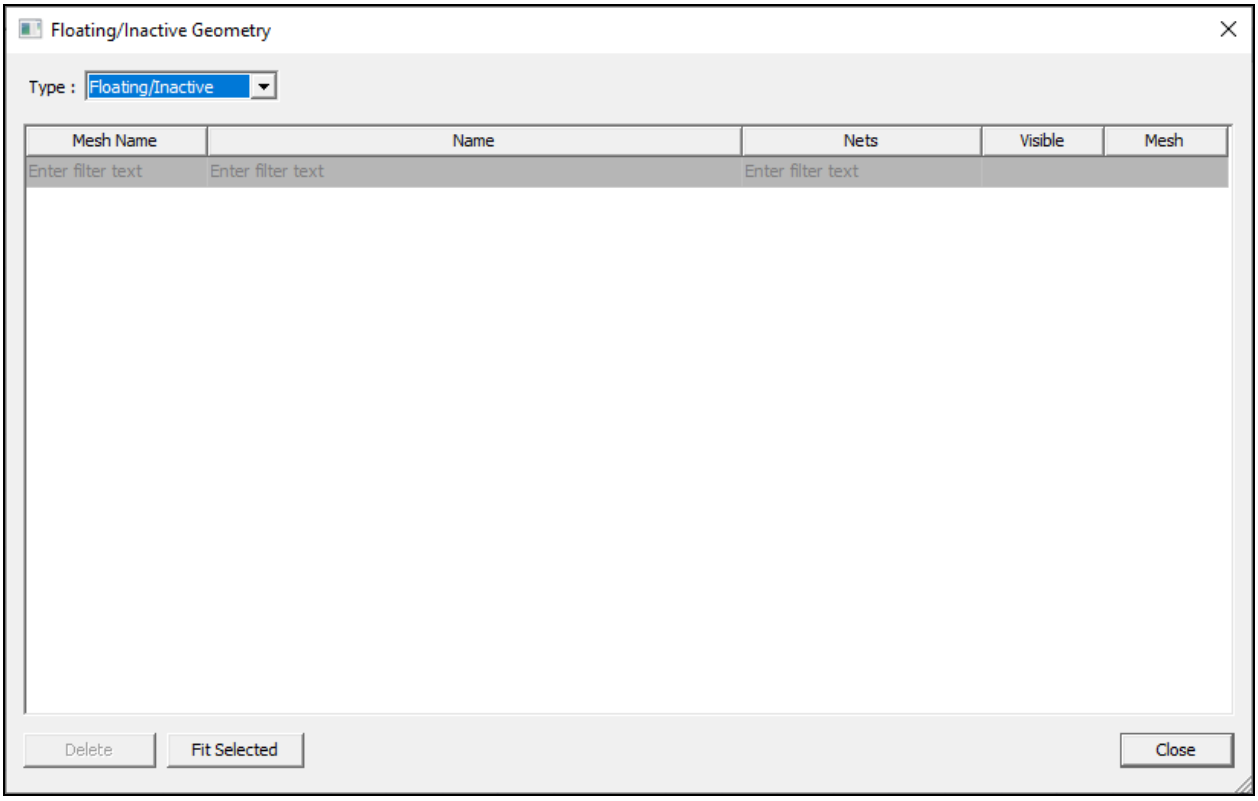
4. From the **Project Manager** window, expand **Project Tree** > **[*Active Design Folder*]** > **Analysis**. Then right-click **HFSS Setup** and select **Pre-Process Geometry** > **Display** to create a surface mesh.

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 **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** and select **HFSS Setup** > **Pre-Process Geometry** > **Floating/Inactive Geometry** to open the **Floating/Inactive Geometry** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Floating/Inactive Geometry window table columns include:

- **Mesh Name** – Name of the mesh body.
- **Name** – Collection of layout primitives comprising the mesh body.
- **Nets** – List of nets assigned to the layout primitives comprising the mesh body.
- **Visible** – Check boxes to toggle visibility in the modeling area.
- **Mesh** – Check boxes to toggle mesh visibility.

1. Select a **Type**:
    a. **Active** shows objects attached to ports or in a Power/Ground net. These are included in the mesh.
    b. **Floating/Inactive** lists objects that are excluded on the mesh.
2. To find an object in the list, enter a search term in the first row.
3. To examine an object to determine why it is active or inactive, select the geometry in the list and click **Fit Selected**. The **Layout Editor** zooms in on the selected geometry.

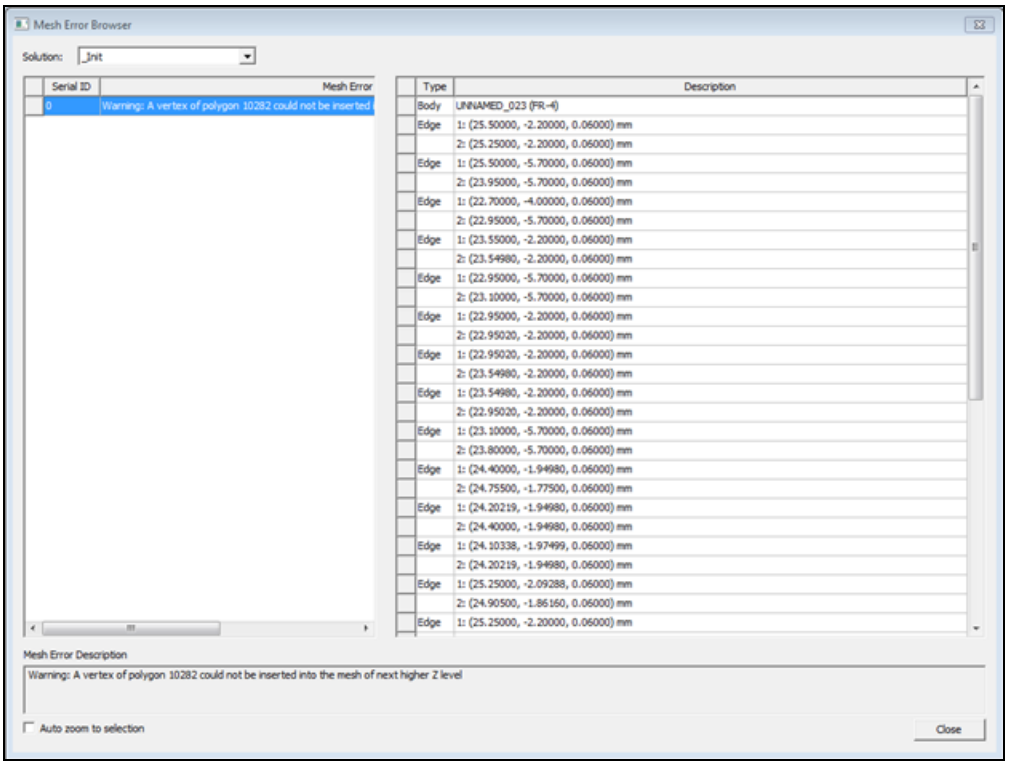
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. To simplify the design by permanently deleting inactive geometry from it, select the inactive geometry in the list and click **Delete**. This is inappropriate as the inactive geometry are filtered out before the simulation is run.
5. To see the mesh of an object, select the geometry and click **Mesh**.
6. To close the window, click **Close**.

> **Note:**
>
> If you change the design, a new mesh is generated when you close and reopen the **Floating/Inactive Geometry** window.

HFSS Mesh Errors

For a given HFSS Setup, in the event of a failure or a difficult mesh, errors, warnings and information may be generated. When available, this feedback can be accessed in the **Project Tree** by right-clicking on the HFSS Setup and choosing **View Mesh Feedback**. This invokes the **Mesh Error Browser** window:



For each **Mesh Error**, a brief description is provided. When a given error is selected in the window, details and dimensional attributes for the contributing bodies are displayed. When one

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or more **Type** entities are selected, a graphical overlay are displayed in the Layout view to better visualize the feedback.

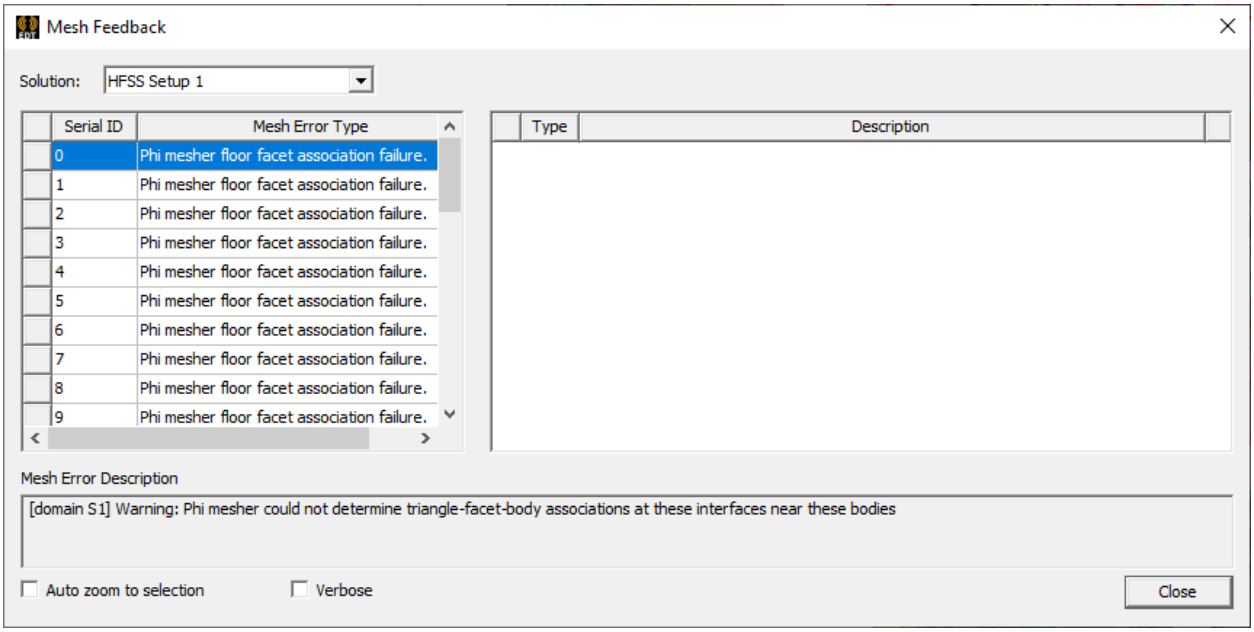
- **Auto zoom to selection**: Enable this setting to dynamically zoom the Layout view based on the error that is selected in the window.

Troubleshooting Mesh Failures

Common causes of mesh failures are improper geometry, incorrect design, or mesh settings. The HFSS mesher can automatically repair and correct a poorly translated and healed model to generate a reliable mesh. The **Profile** reports when mesh repairs are made and the results of these repairs are displayed for every object in the mesh statistics area.

To troubleshoot to correct potential mesh failures on simple geometries:

1. Check the Message Manager for error messages.
2. From the **Project Manager** window, expand the **Project Tree** and **[*Active Design Folder*]**. Then right-click **Analysis** > **HFSS Setup***n*.
3. Right-click and select **View Mesh Feedback** for information on the issue.



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 - HFSS 3D Layout Technical Notes

The basis for the simulation technique that is used to calculate the *S*-parameters and radiated fields for a structure is the mixed potential integral equation (MPIE). Although the MPIE implementation is largely transparent, it helps to have a general understanding of this method to effectively use it in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. The following topics provide an overview of the MPIE method, its implementation, and a description of how S-parameters are computed:

- The MPIE Method
- The Adaptive Analysis Process
- Ansys Fast Solver Technology
- EMI Scanner
- HFSS 3D Layout S-Parameters
- HFSS 3D Layout Modes
- Viewing the HFSS 3D Solution Log File
- Exporting S-Matrices for Full-Wave SPICE Analysis
- Frequency-Dependent Material Properties
- Djordjevic-Sarkar Causal Dielectric Model

The MPIE Method

To generate a solution from which S-parameters can be computed, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform employs the mixed-potential integral equation (MPIE) method. The method of moments (MoM) is applied to the MPIE to solve for ***J***, the current distribution on the 3D surface mesh. The S-parameters and the radiated fields are calculated from ***J***.

In the Ansys Electronics Desktop (AEDT)™ electronics systems design platform, the surface of the geometric model is automatically divided into triangles and rectangles. This collection of triangles and rectangles is referred to as the mesh. A volume mesh is not generated inside the model because at high frequencies the skin depth is so small that a surface mesh is sufficient.

Once the 3D surface mesh has been generated, the following MPIE is applied to it:

$$\hat{n} \times (-j\omega \boldsymbol{A} - \nabla\phi) = \hat{n} \times Z_s \boldsymbol{J}$$

where

- $\hat{n}$ is a unit vector perpendicular to the face of the triangles.
- *j* is the imaginary unit, $\sqrt{-1}$.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

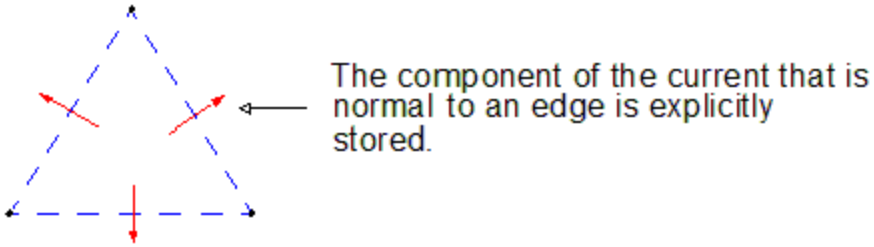
- ω is the angular frequency, $2\pi f$.
- $\boldsymbol{A}$ is the magnetic vector potential.
- ϕ is the electric potential.
- Z_s is the surface impedance.
- $\boldsymbol{J}$ is the current density.

The current normal to the edges of the triangles is calculated.

> **Note:**
>
> The MPIE method is unstable at low frequencies. The Desktop loop tree discretization is a re-ordering of the MoM matrix that provides a highly stable solution at very low frequencies.

Representation of the Current

The value of the current at points inside each basis element (triangle or rectangle) is interpolated on the edges of the basis element. On each basis element, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform stores the components of the current that are normal to the edges of the element. The current inside each element is the superposition of these normal values.



Method of Moments

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform uses the method of moments (MoM) to solve the MPIE and calculate $\boldsymbol{J}$ everywhere on the mesh. The MoM can be broken down into two sections: the basis function and the testing function.

Basis Function

The Desktop uses a zero-order normal element basis function to interpolate the interior current values from values on the edges. Zero-order normal elements have one unknown for each edge in the mesh.

Testing 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 functions are applied to the MPIE to obtain a matrix equation. The matrix equation is used to find ***J***. From ***J***, calculates the S-parameters and the radiated fields. The functions used for the basis functions are also used for the testing functions.

Size of Mesh Versus Accuracy

There are trade-offs among the size of the mesh, the selected level of accuracy, and the amount of available computing resources. The accuracy of the solution depends on the size of the basis elements. Solutions based on meshes that use many small triangles and rectangles are more accurate than solutions based on coarse meshes using few elements. To generate a precise description of the current, each element must occupy a region that is small enough for the current to be adequately interpolated on the normal values. However, meshes with a large number of elements require a significant amount of computing power and memory. Therefore, it is desirable to use a mesh that is fine enough to obtain an accurate current solution but not so fine that it overwhelms the available computer memory and processing power.

To produce an optimal mesh, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform uses an adaptive, or iterative process in which the mesh is automatically refined in critical regions. First, it generates a solution based on a coarse initial mesh. Then, it refines the mesh in critical areas and generates a new solution. When the current and charge distributions on the structure converge to in the selected precision, AEDT breaks out of the loop.

Alternatively, a fixed dense mesh can also be used by choosing the fixed frequency mesh option and specifying a frequency near the high end of the sweep range. The mesh generated using this option is a uniform mesh. The advantage of the fixed frequency mesh over the adaptive mesh is that no extra simulations are needed to generate the mesh before the frequency sweep can be done.

Loop Tree Discretization

The MPIE method is unstable at low frequencies. Each element of the MoM matrix comprises the impedance from an inductive term and a capacitive term. As the frequency moves toward zero, the impedance on the inductive term tends toward zero, while the impedance of the capacitive term tends toward infinity. This leads to numerical instability and poor results because the inductive term is lost in the numerical noise of the capacitive term.This can also be seen by breaking the MoM *Z*-matrix into sub-matrices *S* and *T.* The MoM matrix is

$$Z = S - \omega^2 T$$

where ω is the frequency. The sub-matrices treat the scalar and vector potential parts of the integral equation separately. The elements of the sub-matrices are

$$S_{ij} = \iint (\nabla \cdot f_i) G_\phi (\nabla \cdot f_i) ds\, ds'$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T_{ij} = \iint f_i \cdot G_A \cdot f_j ds ds'$$

where f_i and f_j are the shape or basis functions. As the frequency ω goes to zero, the MoM matrix reduces to *S*. The matrix *S* is nearly singular at very low frequencies; it corresponds to currents with nearly zero divergence.

The Ansys Electronics Desktop (AEDT)™ electronics systems design platform loop tree discretization is a re-ordering of the MoM matrix that explicitly preserves the vector potential part of the MoM matrix that corresponds to the inductance term. It thereby provides a highly stable solution at very low frequencies. At higher frequencies in which the MoM matrix impedances on the inductance and capacitance are on the same order, the loop tree discretization has little or no benefit, but it does not increase computation time or reduce accuracy.

MPIE Method Implementation

To calculate the S-matrix associated with a structure, the Ansys Electronics Desktop (AEDT)™ electronics systems design platform does the following:

- Divides the structure into a 2.5D or 3D surface mesh.
- Computes the currents on each port of the structure that are supported by a transmission line having the same cross-section as the port.
- Computes the full current distribution on the surface of the structure.
- Computes the generalized S-matrix on the amount of reflection and transmission that occurs.

The final result is an S-matrix that allows the magnitude of transmitted and reflected signals to be computed directly from a given set of input signals.

The Adaptive Analysis Process

An adaptive analysis refines the mesh in order to increase the precision and accuracy of the solution. When the Ansys Electronics Desktop (AEDT)™ electronics systems design platform performs an adaptive analysis, the process of computing a current solution is an iterative one. In general, you should first accept the default stopping and the default refinement criteria. Then, after you generate the selected current solutions, check their convergence to see if stopping criteria should be adjusted.

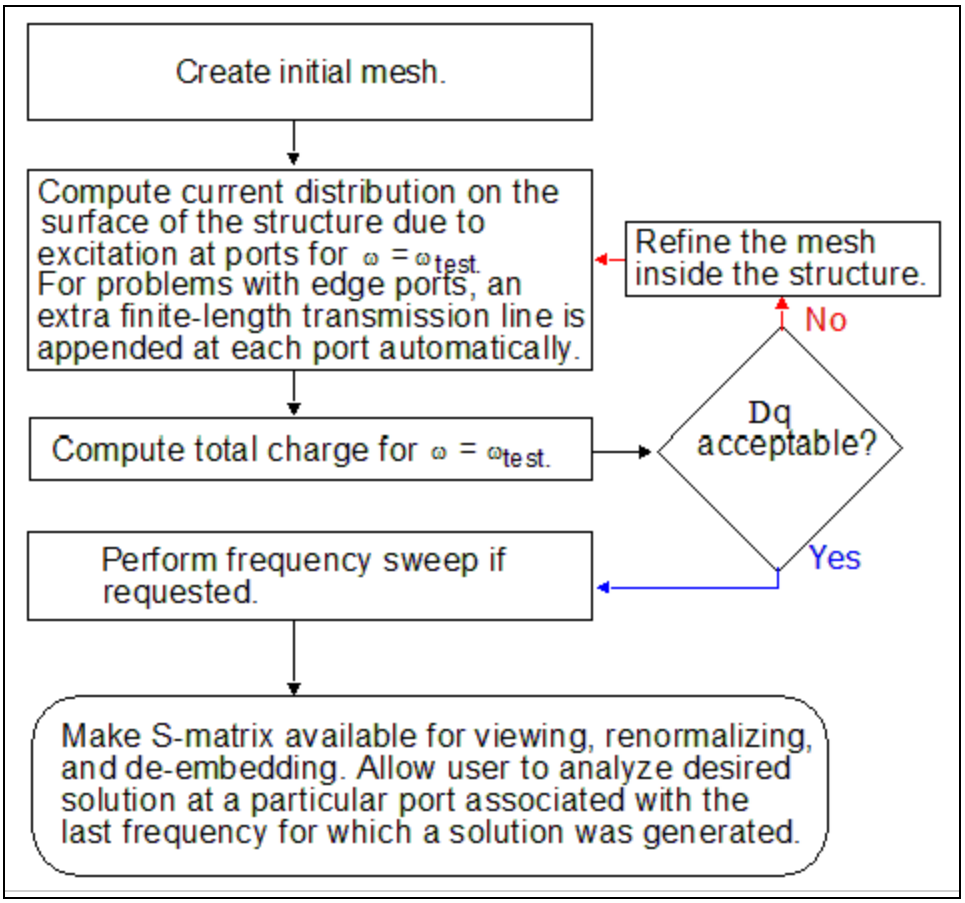
The adaptive frequency is converged when the Delta S for the frequency is reached and all the output variables have converged OR when the maximum number of passes for the frequency is reached. When either of those criteria have been reached, the adaptive refinement moves to the next frequency or stop, if there are no more frequencies. The mesh that has been refined at the end of one adaptive frequency are used as the starting mesh for the next adaptive frequency. The solution is refined in the following way:

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 Desktop generates a dense mesh at a fixed frequency, the frequency specified in the **Frequency** field.
2. The Desktop computes the currents that exist inside the structure when it is excited at a single frequency, the frequency specified in the **Adaptive Frequency** field. If a frequency sweep is being performed, an adaptive analysis is performed only at the frequency specified in this box.
3. A solution is generated on the computed currents.
4. If the number of requested passes is satisfied or the target maximum delta norm is in the selected range, the solution process stops. Otherwise, AEDT performs another iteration.
5. The Desktop refines the mesh in the areas of highest error.
6. Another solution is generated using the refined mesh. AEDT recomputes the error, and the iterative process (solve — error analysis — refine) repeats until the number of requested passes is satisfied or the target maximum delta norm is in the selected range.

If a frequency sweep is being performed, AEDT solves the problem at the other frequency points without further refining the mesh.

The procedure for performing an adaptive solution is 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.

Ansys Fast Solver Technology

Moment method (MoM)-based tools have always required long computation times and large amounts of computer memory. The MoM matrix solution-time typically increases by n^3, where *n* is the number of unknowns, while the MoM matrix memory-requirement increases by n^2. Green's-function calculations and matrix-accumulation times are typically not limiting factors. For practical antenna array analysis, the rapid increase in solution time and memory requirements limits the size of an array that can be studied (to approximately 32 elements).

Ansys MoM Fast Solve technology enables the analysis of larger structures than previously possible. The Ansys Electronics Desktop (AEDT)™ electronics systems design platform uses a new, proprietary algorithm to significantly reduce the memory needed to store and solve the MoM matrix. The algorithm analyzes the electromagnetic characteristics of the structure to determine the different levels of coupling between regions. In many cases, where electromagnetic interaction between regions is low, reduced-order coupling matrices can be employed to accurately capture the electromagnetic effects. The reduced-order coupling matrices save large amounts of memory compared to the full MoM coupling sub-matrices. In many cases, the memory requirements can be reduced by a factor of 15 or more compared to classical MoM methods.

The matrix solution time can also be reduced by employing advanced preconditioning algorithms and iterative solvers on the reduced order MoM matrix.

> **Note:**
>
> The **Ansys MoM Fast Solver** should not be applied to problems with less than 4000 unknowns, or in practice, antenna arrays of 25 or more elements.

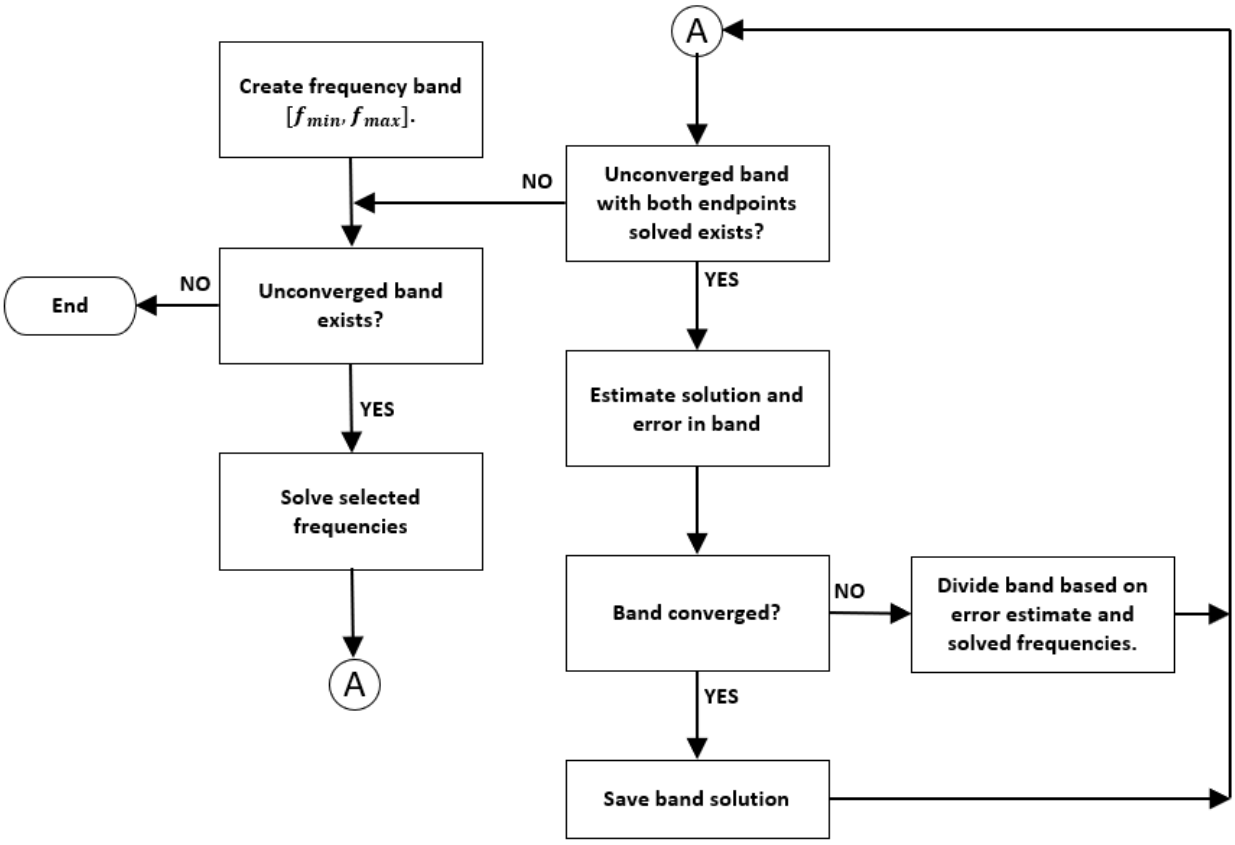
Broadband Fast Frequency Sweeps

A broadband fast sweep estimates the S-parameter solution, and optionally the full-field solution, at every point in the frequency range. The solved frequencies are selected adaptively so that the estimated error across the entire frequency range is less than the specified tolerance. The sweep ends when this tolerance is achieved or the maximum number of frequencies are solved. An accurate broadband response is realized by using not only the value of the solution at the solved frequencies, but also its derivatives with respect to frequency. This requires, however, that all material properties be differentiable at all frequencies (e.g., piecewise-linear properties are not supported).

Choose broadband fast sweep for moderately-sized models when the frequency range is wide and the frequency response is complicated, where interpolating or discrete sweeps would require too many specific frequency points.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Be aware that HFSS uses the finite element mesh refined during an adaptive solution at the solution frequency or the initial mesh generated for the problem if no adaptive solution is created. It uses this mesh without further refinement.



EMI Scanner Technical Notes

These notes aid in running EMI Scanner.

EMI Scanner Rule Setting Notes

Component Pin Grid Settings

Layout tools sometimes define a component's boundary box too large. For rules that check for whether a component falls inside the grid or outside the grid, this can lead to the rule thinking that a component is inside the grid when it is in fact outside the grid (but its boundary box still falls at least partially inside the grid). Users can change how EMI Scanner determines whether the component is inside or outside the grid in one of three ways:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Treat the component as inside the grid if any pin of the component is inside the grid. Use the following settings:

   " Require One Component/Capacitor Pin to be Inside Grid " = yes

   " Require ALL Component/Capacitor Pins to be Inside Grid " = no

2. Treat the component as inside the grid only if all the pins on the component are inside the grid. Use the following settings:

   " Require One Component/Capacitor Pin to be Inside Grid " = yes

   " Require ALL Component/Capacitor Pins to be Inside Grid " = yes

3. Treat the component as inside the grid if the component's boundary box intersects the grid. Use the following settings:

   " Require One Component/Capacitor Pin to be Inside Grid " = no

   " Require ALL Component/Capacitor Pins to be Inside Grid " = yes or no

Decoupling All Plane Combinations

When determining whether to select **Force All Plane Combinations to Be Decoupled**, consider the following stackup.

Assume the net references V02/V03 when on layer S02 and V04/V05 when on S04.

V02 ----------

S02 ------+

V03 -----------|--------

S03 |

V04 -----------|--------

S04 +-----

V05 ----------

For this case, have the tool either look for decaps connecting V02<->V04 and V03<->V05 or have it look for V02<->V04, V02<->V05, V03<->V04, and V03<->V05. By default, it looks for 1st ref layer to 1st ref layer and 2nd ref layer to 2nd ref layer (V02<->V04, V03<->V05). If you want the tool to decouple all ref plane combinations, set this value to YES.

I/O Net Classes

I/O net classes are as follows:

Class 1L – low speed intentional signals that leave the shielded enclosure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Class 1H – high speed intentional signals that leave the shielded enclosure.

Class 2 – signals that leave the board on a cable and remain in the shielded enclosure.

Class 3 – signals that leave the board directly to another board and remain in the shielded enclosure.

HFSS 3D Layout S-Parameters

The S-matrix indicates what fraction of an incident voltage wave is transmitted or reflected at each port. The voltage wave is normalized with respect to the square root of the characteristic impedance of the port (e.g., a = V*Zo-1/2). If the characteristic impedance is completely real, the elements of the S-matrix are equivalent to the square root of the fraction of power transmitted or reflected. An S-matrix for a three-port structure is:

$$\begin{bmatrix} b_1 \\ b_2 \\ b_3 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix}$$

where:

- All quantities are complex numbers.
- a_i represents the incident voltage wave at port i.
- b_i represents the transmitted or reflected voltage wave at port i.
- The phase of a_i and b_i is the phase of the incident and reflected/transmitted voltage wave at t = 0:

  Za_i is the phase angle of the excitation voltage wave on port i at t = 0 (by default, it is zero).

  Zb_i is the phase angle of the reflected or transmitted voltage wave with respect to the excitation voltage wave.

- S_{ij} is the S-parameter describing how much of the incident voltage wave at port j is reflected back or transmitted to port i (e.g., S_{31} is used to compute the amount of power on the port 1 excitation that is transmitted to port 3. The phase of S_{31} specifies the phase shift that occurs as the voltage wave travels from port 1 to port 3.).

S-Parameters and Frequency Points

The S-parameters associated with a structure are a function of frequency. Separate solutions and S-matrices are generated for each frequency point 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.

Computing Characteristic Impedance

The Desktop must calculate the characteristic impedance of each port in order to compute a renormalized S-matrix.

The characteristic impedance of microstrip and stripline ports is calculated using a variation of the voltage-current definition of impedance

$$Z_c = (V,I)/I^2,$$

where the voltage *(V,I)* is a weighted average of the voltage on the strip defined as

$$(V,I) = -\int_{strip} J(l_{strip}) \left(\int_{ground}^{strip} E(l_{strip}) \cdot dl \right) dl_{strip}.$$

Since the path of the integral of the electric field is not path-independent for time-varying fields, the voltage on the strip varies with position. This definition uniquely defines a voltage.

For coupled microstrip and stripline ports, the characteristic impedance is defined as power over the square of the current. The power is defined as

$$P = \iint_S \hat{I} \cdot (E \times H^*),$$

$$\hat{I}$$

The definition of characteristic impedance for coplanar waveguides and slotline ports is defined as the square of the voltage over power. The power definition is the same for coupled ports, and the voltage is defined as the integral of the electric field across a slot.

Renormalized S-Matrices

Before a generalized S-matrix can be used to compute the reflection and transmission of signals, the generalized S-matrix must be normalized to the selected impedance. For example, if a generalized S-matrix has been normalized to 50 ohms, it can be used to compute reflection/transmission directly from signals normalized to 50 ohms:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} Vo_1 \\ Vo_2 \\ Vo_3 \end{bmatrix} = [S_{50}] \begin{bmatrix} Vi_1 \\ Vi_2 \\ Vi_3 \end{bmatrix}$$

where:

- The input signals, Vi_i, and output signals, Vo_i, are both normalized to 50 ohms.
- To renormalize an S-matrix to a specific impedance, the system calculates a unique impedance matrix associated with the structure.

The unique impedance matrix, Z, is:

$$Z = \sqrt{Z_0}(I+S)(I-S)^{-1}\sqrt{Z_0},$$

where:

- *S* is the *nxn* generalized S-matrix.
- *I* is an *nxn* identity matrix.
- Z_0 is a matrix with the characteristic impedance of each port as a diagonal value.

The renormalized S-matrix is then calculated from *Z* using this relationship:

$$S_\Omega = \sqrt{Y_\Omega}(Z-Z_\Omega)(Z+Z_\Omega)^{-1}\sqrt{Z_\Omega},$$

where:

- *Z* is the structure's unique impedance matrix.
- Z_W and Y_W are matrices with the selected impedance and admittance as diagonal values.

Visualize the generalized S-matrix as an S-matrix that has been renormalized to the characteristic impedances of the structure. Therefore, if a diagonal matrix containing the characteristic impedances of the structure is used as Z_W in the previous equation, the result is the generalized S-matrix again.

> **Note:**
>
> S-matrices for modes other than the dominant one cannot be renormalized or used in circuit simulators.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Computing the Z-Matrix

The impedance matrix, Z, is calculated on the S-matrix as follows:

$$Z = \sqrt{Z_0}(I-S)^{-1}(I+S)\sqrt{Z_0}$$

where:

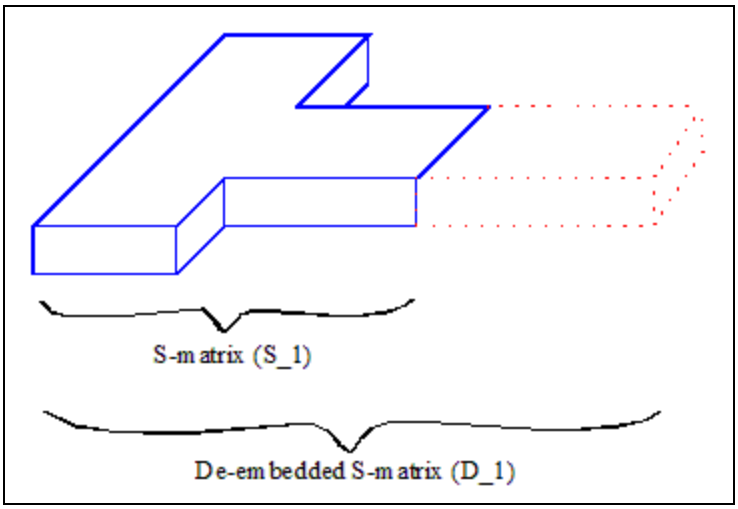
- *S* is the *nxn* generalized S-matrix.
- *I* is an *nxn* identity matrix.
- Z_0 is a diagonal matrix having the characteristic impedance (Z_0) of each port as a diagonal value.

Computing the Y-Matrix

The admittance matrix, Y, is the inverse of the impedance matrix, Z.

Deembedded S-Matrices

Deembedding allows you to shift the reference plane of the S-matrix for any port by adding (or subtracting) a length of lossless transmission line to (or from) a port. The following figure illustrates such a case.



The S-matrix, S_1, describes the behavior of the structure that was modeled.

The deembedded S-matrix, D_1, describes the behavior of a structure created by subtracting or adding a length of lossless transmission line to the structure that was modeled. It is computed by

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

phase rotating the S-matrix inward toward or outward away on the structure by the specified length at each port.

To shift the reference plane of the structure, the S-matrix of the modified structure can be calculated using the following relationship:

$$[S'] = [e^{\gamma l}][S][e^{\gamma l}]$$

Where: $[e^{\gamma l}]$

is a diagonal matrix with the following entries:

$$\begin{bmatrix} e^{\gamma_1 l_1} & 0 & 0 \\ 0 & e^{\gamma_2 l_2} & 0 \\ 0 & 0 & e^{\gamma_3 l_3} \end{bmatrix}$$

- $\gamma = \alpha + j\beta$ is the complex propagation constant, where α is the attenuation constant of the current and β is the phase constant associated with the uniform transmission line at port i.
- l_i is the length of the reference plane shift. A positive value indicates a shift toward the structure by a length *l*.

The value of $\gamma$for each port is automatically calculated by the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. However, when performing deembedding, the attenuation constant, α, at each edge port is ignored. Therefore, only the phase of the S-parameter is affected.

HFSS 3D Layout Modes

For a transmission line with a given cross-section, there is a series of basic current patterns (modes) that satisfy Maxwell's equations at a specific frequency. Any linear combination of these modes can exist on the transmission line. By default, the system computes only the dominant mode current pattern.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modes, Reflections, and Propagation

It is possible for a current solution generated by an excitation signal at one specific mode to contain reflections of higher-order modes which arise due to discontinuities in the high frequency structure. If these higher-order modes are reflected back to the excitation port or transmitted onto another port, the S-parameters associated with these modes should be calculated.

If the higher-order mode decays before reaching any port — either because of attenuation due to losses or because it is a non-propagating evanescent mode — there is no need to obtain the S-parameters for that mode. Therefore, one way to avoid the need for computing the S-parameters for a higher-order mode is to include a length of transmission line in the geometric model that is long enough for the higher order mode to decay.

For best results, include a small length of the transmission line that is connected to each port in the model of the structure. For example, if one port of a structure is connected to a microstrip line, the part of the geometric model leading up to the port must have the cross-section of the microstrip line.

Modes and Frequency

The current patterns associated with each mode vary with frequency, as do the propagation constants and impedances. When performing frequency sweeps, be aware that as the frequency increases, the likelihood of higher order modes becoming propagating modes also increases.

Evanescent Modes and Attenuation

It is possible for a current solution generated by an excitation signal at one specific mode to contain reflections consisting of higher-order modes.

Viewing the HFSS 3D Solution Log File

After a simulation has completed, HFSS 3D prints messages about the success or failure of the analysis to the Ansys Electronics Desktop Message Manager window. HFSS 3D also writes details regarding the simulation to a solution log file. The log file information may be useful for diagnosing why a particular analysis failed.

- From a schematic design, right-click the analysis setup under the Analysis icon in the **Project Tree** and select **Browse Solution Log File**.
- From a netlist design, right-click the circuit icon in the **Project Tree** and select **Browse Solution Log File** to open the log file in a standard Notepad window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

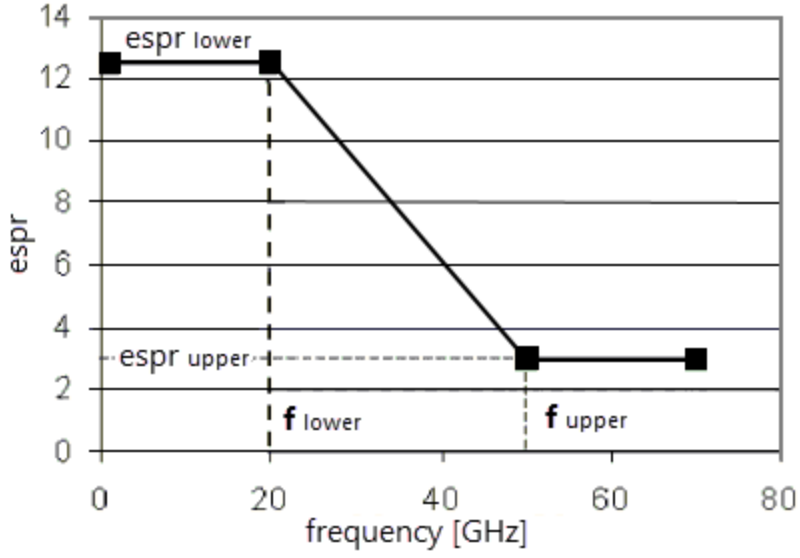
Exporting S-Matrices for Full-Wave SPICE Analysis

Significant coupling, that occurs in closely located circuits reaching frequencies above one GHz, can be included in circuit analysis. When S-parameter data calculated by the Ansys Electronics Desktop (AEDT)™ electronics systems design platform is exported to PSpice, HSpice, and Maxwell Spice software, transformation data between voltages and currents is constructed. This transformation data is based on the S-parameter data; therefore full-wave effects can be modeled in the circuit analysis.

When S-parameter data is exported from AEDT to PSpice or HSpice, AEDT creates an equivalent circuit. The circuit is represented by an N-port with input and output terminals. It includes the same frequency response behavior as the original device analyzed in AEDT. Frequency domain information from AEDT is transferred to circuit simulators that operate in the time domain. Broadband matching of frequency responses is ensured while the full-wave solution accuracy is preserved. When S-parameter data is exported from AEDT to Ansys's Maxwell Spice software, AEDT creates a new "full-wave *N*-port subcircuit". This subcircuit is also represented by an N-port that includes the same frequency response behavior as the original device.

Frequency-Dependent Material Properties

The properties of some materials vary with the frequency of the field excitation. Frequency dependence is often linear within a certain range and is constant outside the frequency range, as shown in the following figure.



- $epsr_{lower}$ is the relative permittivity of a material below the frequency range 20 - 50 GHz.
- $epsr_{upper}$ is the relative permittivity of a material above the frequency range 20 - 50 GHz.
- f_{lower} is the lower frequency, below which the material property is constant.
- f_{upper} is the upper frequency, above which the material property is constant.

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, to account for this variance within a given frequency range, use the **Piecewise Linear Material Input** window to specify a property's values at frequencies below and above the frequency range. Based on these values, Ansys Desktop automatically creates a linear dataset that specifies the property's values at the selected frequencies during solution generation. This dataset can be modified with additional points if selected.

Obtaining Broadband Matching between Huray and Hammerstad Models

This topic contains the procedure for obtaining broadband matching between Huray and Hammerstad models.

> **Note:**
>
> Before reading this topic, see the sections Huray Surface Roughness Model and Modified Hammerstad Model for the definitions of Scale Factor (SF) and to see how each of these models is expressed as a function of frequency.

The simplest way to create a Hammerstad model that is in some sense close to a Huray model is to ensure the maximum values of the models are the same.

$$SF = 1 + \frac{3}{2}(SR)$$

$$SR = \frac{2}{3}(SF - 1)$$

Then, try to line up the models at their 50% crossing points. For the Hammerstad model, that occurs when (1):

$$1 + \left(\frac{2}{\pi}\right)(SF - 1)arc\tan\left(\frac{f}{f_{HJ}}\right) = \frac{1+SF}{2}$$

$$f = f_{HJ}$$

For the Huray model, the 50% point is reached when (2):

$$\frac{\frac{f}{f_{HH}}}{\frac{f}{f_{HH}} + \sqrt{\frac{f}{f_{HH}}} + \frac{1}{2}} = \frac{1}{2}$$

By solving for *f*, the equation reduces to the following (3):

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f = f_{HH}\left(1 + \frac{\sqrt{3}}{2}\right)$$

Therefore, for 50% of points to agree, the following condition must be realized (4):

$$f_{HJ} = f_{HH}\left(1 + \frac{\sqrt{3}}{2}\right) \cong 1.866 f_{HH}$$

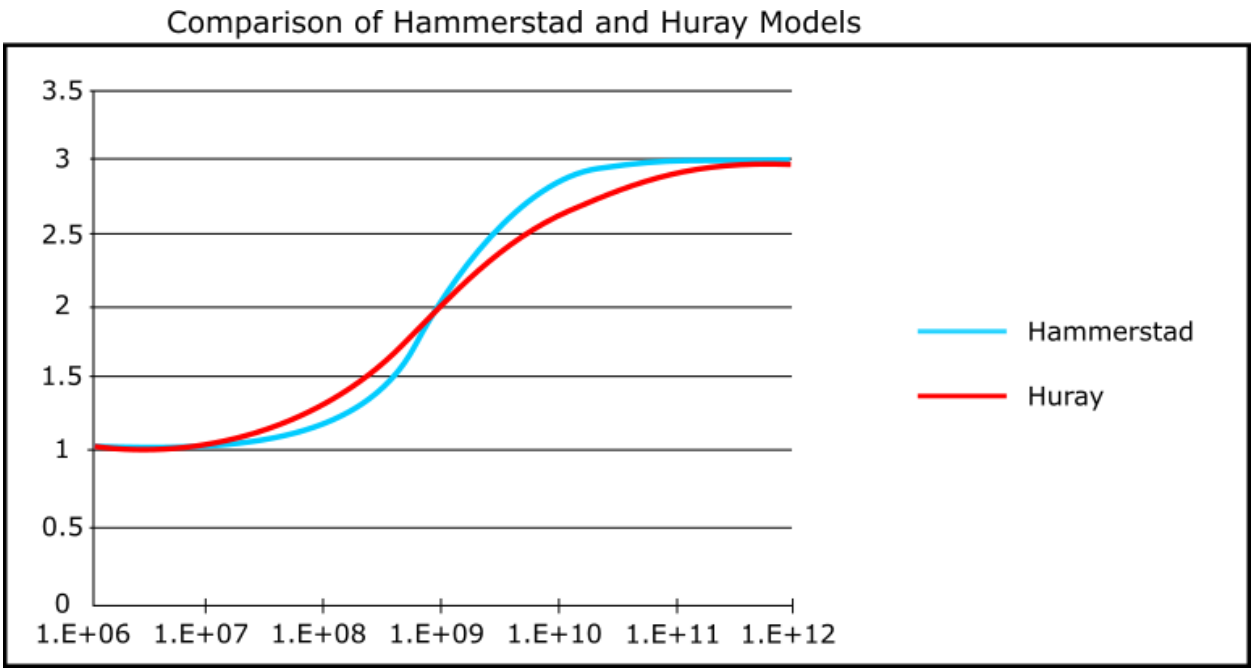
Using equation 4 and relating back to the model parameters, the following is obtained (5):

$$\frac{1}{1.4\pi\Delta^2\mu\delta} = \frac{1.866}{\pi a^2 \mu\sigma}$$

Compare the two models. For the Hammerstad model, set *SF*=3 and Δ=1.766m and let the material be copper. This conveniently yields f_{HJ} = 1 GHz.

Using the preceding relations, the corresponding Huray model has *SR*=4/3 and *a*=2.854µm.

Plots of $H_{HJ}(f)$ and $H_{HH}(f)$ are shown in the following figure.



The figure demonstrates the expected match between the two curves at the 50% point (1 GHz), and the expected match at the low and high frequency extremes. It is also clear the Hammerstad model transitions between low and high frequency more rapidly than the Huray model. This is inherent to the mathematical functions used in the model. Since the Huray model is based on the physics of the underlying loss mechanism while the Hammerstad model is a curve fit, it is reasonable to expect the Huray model to be more realistic.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The preceding model-matching procedure potentially may be critiqued as follows:

- The matching is exact at some (three) points and is decent across the entire frequency range.
- The computed snowball radius is actually greater than the RMS surface roughness value. This does not seem realistic.

An alternative procedure is to try matching the two models at low frequencies. The rationale for doing this is the loss measurements used to derive a Hammerstad model are likely to be taken mainly at frequencies well beneath the 50% point. Therefore, the higher frequency values might not be given as much weight in the matching process. This approach is described below.

Obtaining Better Low Frequency Matching

The low frequency match could be improved by shifting the Huray model curve to the right. Such a shift can be accomplished by reducing the value of the snowball radius; this increases the Huray corner frequency f_{HH}.

No unique method exists to do this since there are many different frequency ranges and error metrics that could be used. Results obtained from some trial and error experiments reveal that a good starting point is using the broadband parameter values:

$$SR = \frac{2}{3}(SF - 1)$$

A match qualitatively similar to that shown in the following figure is obtained by choosing:

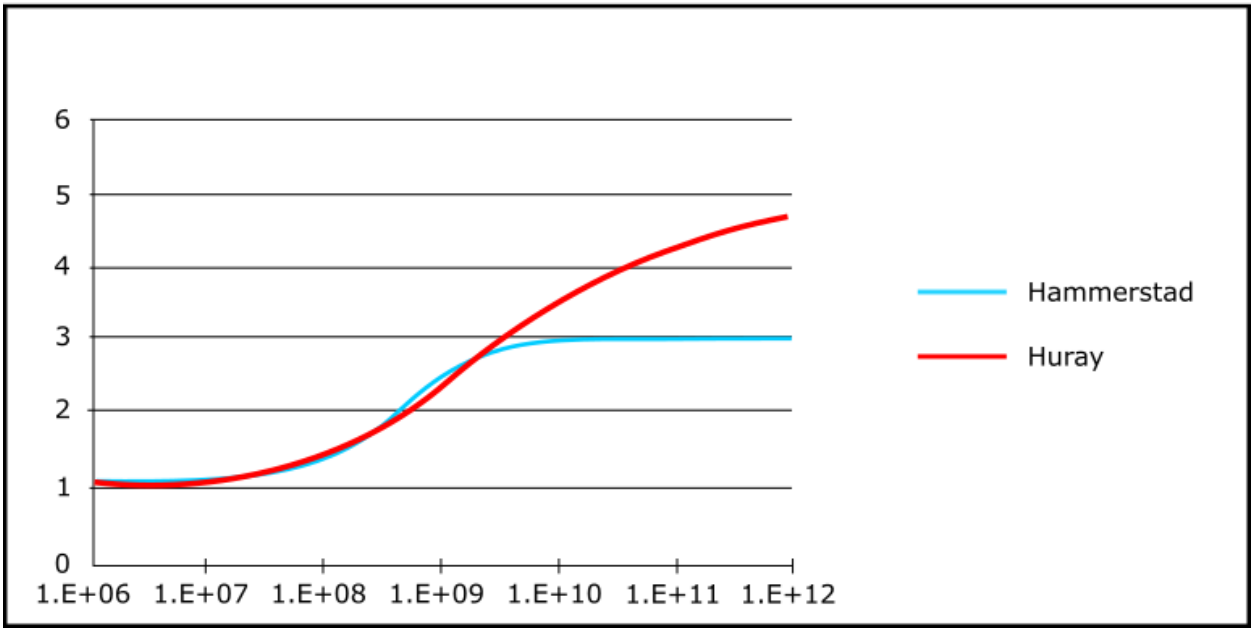
$$a = 0.65\Delta \quad SR = \frac{4}{3}(SF - 1)$$

For more information, see references below.

Setting a = 1.148μm and SR = 2.667 results in the following comparison:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Comparison of Hammerstad and Huray Models with Low-Frequency Matching Emphasized at the Expense of High-Frequency



The match between the two is very good up to about 4 GHz, well beyond the 50% crossing point of the Hammerstad model at 1 GHz. Beyond this frequency, the fitted Huray model predicts significantly higher losses (66% greater) than the Hammerstad model. This is the price paid for the improved low frequency match. Of course it was assumed the Hammerstad model may not be accurate in this higher frequency range anyway.

To summarize, the Hammerstad model is fundamentally different from the Huray model. Since Hammerstad is strictly a mathematical model and the Huray model is a physics-based model, *the conversion between the two is not exact*. An approximate conversion is indeed possible, as shown, and the choice of conversion process depends on whether a broadband or a low-pass match is selected. Two conversion schemes have been described here, but a continuum of different choices is available.

Expect that any usable conversion scheme has Huray model parameters that satisfy the following bounds:

$$0.65\Delta \leq a \leq 2\Delta$$

and

$$\frac{2}{3}\left(SF-1\right) \leq SR \leq \frac{3}{2}\left(SF-1\right)$$

References:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. E. Bogatin, D. DeGroot, P.G. Huray, and Y. Shlepnev. "Which one is better? Comparing options to describe frequency dependent losses." *DesignCon, 2013*.

Modified Hammerstad Model

In Ansys Electronics Desktop, the Hammerstad-Jensen surface roughness model is only implemented for metal layers in SIwave designs and for Finite Conductivity boundaries assigned to applicable edges in 2D Extractor designs. For HFSS and HFSS 3D Layout designs, Hammerstad surface roughness can be mapped to the Huray model when assigning Finite Conductivity boundaries.

This section describes a modified Hammerstad model and how it is expressed as a function of frequency. The Hammerstad model is strictly mathematical. As mentioned above, it is possible to map the Hammerstad surface roughness data to a physics-based Huray model.

The modified Hammerstad model is described by two parameters, the RMS surface roughness Δ and the scale factor *SF*. In the original Hammerstad-Jensen model, the value of *SF* = 2.

The model equation is as follows:

$$\mathrm{H_{HJ}}\left(f\right) = 1 + \left(\frac{2}{\pi}\right)\left(SF - 1\right)arc\tan\left[1.4\left(\frac{\Delta}{\delta(f)}\right)^2\right]$$

$\delta\left(f\right)$ is given by

$$\delta\left(f\right) = \frac{1}{\sqrt{\pi f \mu \sigma}}$$

Therefore, the model equation reduces to:

$$\mathrm{H_{HJ}}\left(f\right) = 1 + \left(\frac{2}{\pi}\right)\left(SF - 1\right)arc\tan\left[1.4\pi\Delta^2\mu\sigma f\right]$$

The model's corner frequency f_{HJ} is defined using the formula:

$$f_{HJ} = \frac{1}{1.4\pi\Delta^2\mu\sigma}$$

Therefore, the modified Hammerstad model can be written as:

$$\mathrm{H_{HJ}}\left(f\right) = 1 + \left(\frac{2}{\pi}\right)\left(SF - 1\right)arc\tan\left[\frac{f}{f_{HJ}}\right]$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> At low frequencies, the value of $H_{HJ}(f)$ approaches 1. At high frequencies, the value approaches *SF*.

Additional Reading:

1. E. Bogatin, D. DeGroot, P.G. Huray, and Y. Shlepnev. "Which one is better? Comparing options to describe frequency dependent losses." *DesignCon, 2013*.

Djordjevic-Sarkar Causal Dielectric Model

From the paper "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality" (A. R. Djordjevic, R. D. Biljic, V. D. Likar-Smiljani, and T. K. Sarkar, *IEEE Trans. on Electromagnetic Compatibility*, Nov. 2001, p. 662), the following simple analytic model was proposed for the frequency dependence of the complex relative permittivity in FR-4 dielectric materials:

$$\varepsilon(\omega) = \varepsilon'(\omega) + j\varepsilon''(\omega) = \varepsilon_\infty + \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} \ln\left(\frac{\omega_B + j\omega}{\omega_A + j\omega}\right) + \frac{\sigma}{j\omega\varepsilon_0} \qquad (1)$$

where

ε_∞ = high frequency ("optical") relative permittivity

$\Delta\varepsilon = \varepsilon_{DC} - \varepsilon_\infty$ = difference between the DC and optical permittivity

σ_{DC} = DC conductivity, typically very close to 0 in FR-4

ε_0 = permittivity of free space $(8.8542\times10^{-12}\text{F/m})$

$\omega = 2\pi f$ = angular frequency

ω_A = lower (angular) transition frequency; below this frequency the permittivity quickly approaches its DC value

ω_B = upper (angular) transition frequency; above this frequency the permittivity quickly approaches its high-frequency (optical) limit

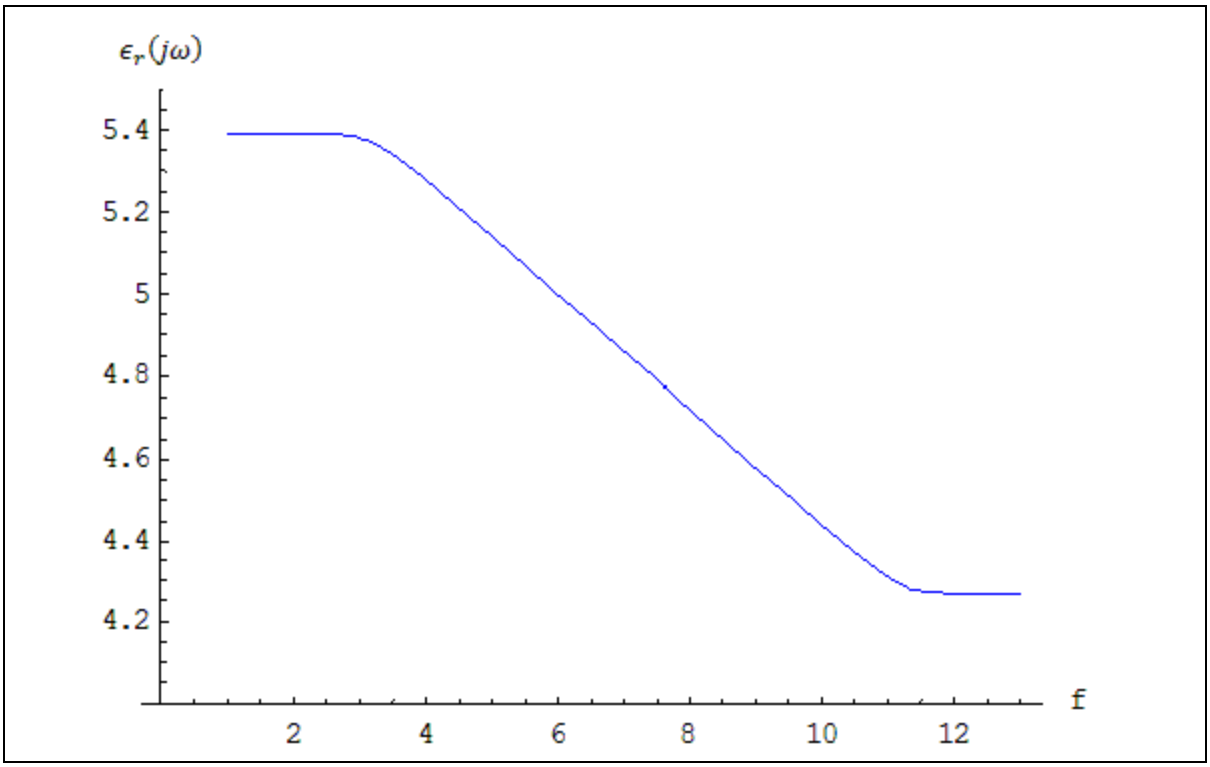
This model is an analytic function and, hence, causal in the time domain. It is very simple and only five parameters define it.

The loss tangent is 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.

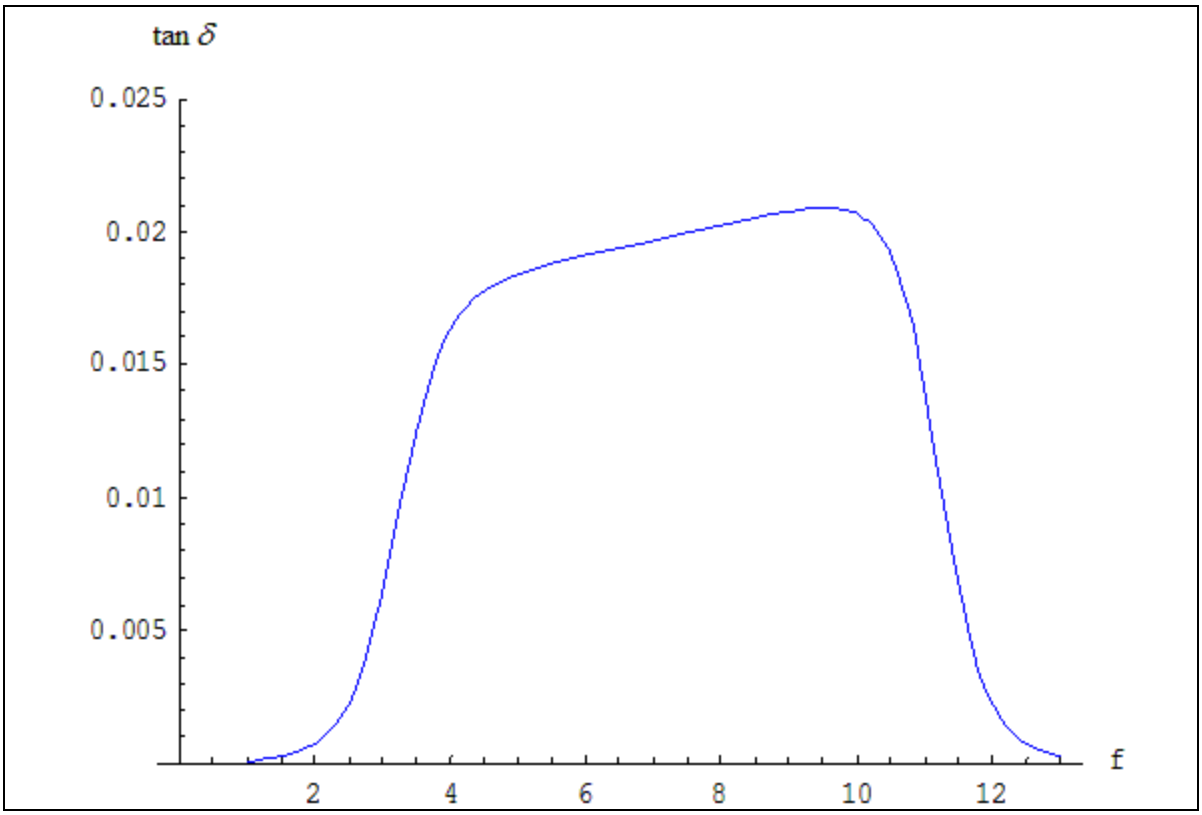
$$\tan\delta = -\frac{\varepsilon''(\omega)}{\varepsilon'(\omega)}$$

The behavior of the real part of permittivity versus frequency using the Djordjevic-Sarkar model is given by way of the following plot:



The loss tangent predicted by the model (for the case $\sigma = 0$) is shown in the following diagram.

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 both curves have two transition frequencies, the first around 10^3 Hz and the second around 10^{11} Hz.

These correspond to the values $\omega_A = 10^4 \text{rad/s}$ and $\omega_B = 10^{12} \text{rad/s}$ used in the model.

The lower transition frequency is evident in the experimental data for FR-4; the upper transition frequency is best thought of as a fitting parameter. Measured data for FR-4 up to 10 GHz does not show the upper transition frequency. Typically, the upper transition frequency is placed at very high frequency, beyond the limits of practical interest.

Another major feature of the model is the loss tangent is approximately constant between the two transition frequencies. This constant loss tangent behavior is also observed experimentally in FR-4, except at low frequencies where it decreases. At very low frequency the loss tangent starts to rise again due to nonzero DC conductivity.

Djordjevic-Sarkar Model Parameter Calculation

The model parameters can be calculated if the following information about the material is known:

- DC conductivity, σ_{DC} (optional; default value is 0)
- DC permittivity, ε_{DC} (optional; you have to select the default.)
- A triplet of real numbers, consisting of
- The measurement frequency ω_1

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 permittivity ε_1 at this frequency
- The loss tangent tan δ_1 at this frequency

From the model equation, observe that

$$\varepsilon(\omega_1) = \varepsilon_\infty + \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} \ln\left(\frac{\omega_B + j\omega_1}{\omega_A + j\omega_1}\right) + \frac{\sigma_{DC}}{j\omega_1\varepsilon_0}$$

$$\cong \varepsilon_\infty + \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} \ln\left(\frac{\omega_B + j\omega_1}{j\omega_1}\right) + \frac{\sigma_{DC}}{j\omega_1\varepsilon_0} \quad (3)$$

$$= \varepsilon_\infty + \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} \ln\left(\frac{\sqrt{\omega_B^2 + \omega_1^2}}{\omega_1}\right) - j\left[\frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} \cdot \tan^{-1}\left(\frac{\omega_B}{\omega_1}\right) + \frac{\sigma_{DC}}{\omega_1\varepsilon_0}\right]$$

Knowing the imaginary part of $\varepsilon(\omega_1) = -\varepsilon_1 . \tan\delta_1$. Therefore getting,

$$\varepsilon_1 \tan\delta_1 = \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} \cdot \tan^{-1}\left(\frac{\omega_B}{\omega_1}\right) + \frac{\sigma_{DC}}{\omega_1\varepsilon_0} \quad (4)$$

$$K = \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} = \frac{\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1\varepsilon_0}}{\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right)} \quad (5)$$

The following equation for the real part also holds:

$$\varepsilon_1 = \varepsilon_\infty + K \ln\left(\frac{\sqrt{\omega_B^2 + \omega_1^2}}{\omega_1}\right) \quad (6)$$

Inserting this expression allows us to solve for the high-frequency permittivity:

$$\varepsilon_\infty = \varepsilon_1 - K \ln\left(\frac{\sqrt{\omega_B^2 + \omega_1^2}}{\omega_1}\right) \quad (7)$$

If you know the DC value of real permittivity, you can find $\Delta\varepsilon = \varepsilon_{DC} - \varepsilon_\infty$ and use the above expression to calculate the lower transition frequency.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\omega_A = \frac{\omega_B}{\exp\left(\frac{\Delta\varepsilon}{K}\right)} \tag{8}$$

Check the computed value of lower transition frequency to ensure that it is much less than the measurement frequency. If not, raise the DC permittivity; this reduces the value of the lower transition frequency. If you do not know the DC permittivity value, you may select one yourself to satisfy the condition:

$$\omega_A << \omega_1$$

A simple heuristic is to choose $\Delta\varepsilon = 10 \cdot \tan\delta_1 \cdot \varepsilon_\infty$.

Expressions for Permittivity and Conductivity

The final expressions are as follows:

Relative Permittivity:

$$\varepsilon(f) = \varepsilon_\infty + \frac{K}{2}\ln\left(\frac{f_B^2 + f^2}{f_A^2 + f^2}\right) \tag{9}$$

Conductivity:

$$\sigma(f) = \sigma_{DC} + 2\pi f\varepsilon_0 K \cdot \left[\tan^{-1}\left(\frac{f}{f_A}\right) - \tan^{-1}\left(\frac{f}{f_B}\right)\right] \tag{10}$$

Here $f_A = \omega_A/2\pi$ and $f_B = \omega_B/2\pi$ is the frequency in Hz.

In the above expressions:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f_B = 10^{12} / 2\pi \tag{11}$$

$$K = \frac{\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0}}{\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right)} \tag{12}$$

$$\varepsilon_\infty = \varepsilon_1 - \frac{1}{2} K \ln\left[\left(\frac{\omega_B}{\omega_1}\right)^2 + 1\right] \tag{13}$$

$\Delta\varepsilon = \varepsilon_\infty - \varepsilon_{DC}$ if the DC permittivity is known; else $\Delta\varepsilon = 10 \cdot \tan\delta_1 \cdot \varepsilon_\infty$

$$f_A = \frac{f_B}{\exp\left(\frac{\Delta\varepsilon}{K}\right)} \tag{14}$$

Check the Limit for Input Parameters

The slope parameter K must be > 0. Therefore you must have

$$\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0} > 0 \tag{15}$$

This can occur if you enter a DC conductivity higher than the effective conductivity at the measurement frequency. This is impossible in the Djordjevic model, because its conductivity increases monotonically with frequency. It will also happen if the measurement frequency is too low (e.g., Hz instead of GHz) or if the permittivity value is unreasonably low.

For the model to be passive, we must have $\varepsilon_\infty \geq 1$

Therefore from (5) and (7) the following is true.

$$\varepsilon_\infty = \varepsilon_1 - \frac{\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0}}{2\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right)} \ln\left[\left(\frac{\omega_B}{\omega_1}\right)^2 + 1\right] \geq 1 \tag{16}$$

Define the quantity

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\tan\delta_{\max} = \frac{2\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right)}{\ln\left[\left(\frac{\omega_B}{\omega_1}\right)^2 + 1\right]} \tag{17}$$

Using this in 16 simplifies the expression as follows.

$$\varepsilon_1 \cdot \tan\delta_{\max} - \left(\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1\varepsilon_0}\right) \geq \tan\delta_{\max} \tag{18}$$

Then, the condition for passivity can be written as follows:

$$\varepsilon_1 \cdot \left(\tan\delta_{\max} - \tan\delta_1\right) \geq \tan\delta_{\max} - \frac{\sigma_{DC}}{\omega_1\varepsilon_0} \tag{19}$$

For this to hold, with $\varepsilon > 0$, the following two conditions are appropriate

$$\tan\delta_{\max} - \tan\delta_1 > 0 \quad \text{or} \quad \tan\delta_1 < \tan\delta_{\max} \tag{20}$$

$$\varepsilon_1 \geq \varepsilon_{1,\min} = \frac{\tan\delta_{\max} - \frac{\sigma_{DC}}{\omega_1\varepsilon_0}}{\tan\delta_{\max} - \tan\delta_1} \tag{21}$$

You see on the expression for tan δ_{max} given in (17) the maximum loss tangent is a function of only the upper corner frequency and the measurement frequency.

Reducing the upper corner frequency increases the value of tan δ_{max} at least as long as the following condition is realized.

ω_B is much greater than ω_1

But reducing the upper corner frequency is not very attractive, because it has not been observed in experimental data. Therefore it is an upper bound on the loss tangents the Djordjevic-Sarkar model can handle. Equation (21) indicates that if the permittivity is too low, the optical permittivity goes below 1. If this condition is violated, two possible fixes are to raise the value of ε or to reduce the measurement frequency. But by the same argument as before, reducing the upper corner frequency is not attractive and it is the minimum value of the permittivity. If you reduce the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value of the measured loss tangent; this makes the denominator in (21) larger and lowers the value of $\varepsilon_{1,min}$.

For more information, see Djordjevic-Sarkar Model Parameter Calculation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19 - 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.

20 - 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.

CPLD

See Complex Programmable Logic Device.

CRC

See Cyclic Redundancy Check.

CSIC

See Customer-Specific Integrated Circuit.

Customer-Specific Integrated Circuit (CSIC)

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

dBm

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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)

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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)

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 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

(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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

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 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

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

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 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

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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)

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Glossary: I

IBIS

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

(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.

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

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)

A shift register whose data input is generated as an XOR or XNOR of two or more elements in the register chain.

Linearity

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Unit qualifier (symbol = u) representing one millionth, or 10-6. For example, 3uS stands for 3 x 10-6 Seconds.

Microwave

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 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)

A generic name for a group of advanced interconnection and packaging technologies featuring unpackaged integrated circuits mounted directly onto a common substrate.

Multilayer

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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)

Nanophase Materials

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 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.

NMOS

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

A memory device which does not lose its data when power is removed from 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.

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

.

.amat file 11-124

.NET assemblies, additional 14-565

.NET classes, Python scripts 14-556

.NET interfaces, Python scripts 14-556

#

2D and Circuit

- 2D Overlaying Far Fields on a 3D View 14-478
- 2D Overlaying Near Fields on a 3D View 14-480
- 2D Overlaying Surface Currents on a 3D View 14-476
- 3D Polar Plot 14-446
- 3D Rectangular Plot 14-445
- Adding a New Output Variable 14-536
- Building an Expression Using Existing Quantities 14-536
- Creating a Report from a File 14-457
- Data Table 14-442
- directive gain 14-515
- Display Types 14-438
- Eye Diagram Plot 14-448
- modifying report data 14-465
- modifying reports 14-465
- New Report window 14-434
- Plot-On-Schematic 14-454
- Polar Plot 14-441
- Radiation Efficiency 14-515
- Rectangular Plot 14-439
- Rectangular Stacked Plot 14-444
- Rectangular Contour Plot 14-447
- Report Types 14-433
- Reports With Differential Pairs 14-456
- Selecting a Far-Field Quantity to Plot 14-514
- Selecting a Function 14-507
- Selecting the Matrix Display Format 14-539
- Smith Chart 14-443
- Smith Contour Plot 14-448
- Spinning a 3D Report 14-475
- Stacked Eye Diagram Plot 14-449
- Statistical Eye Plot 14-451
- Updating post-processing data 14-484

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Frequency Animation 14-540

2D model files, importing 5-5, 5-5

2D Phase Animation 14-543

3D

push and pop coordinate systems 7-359, 7-366

structures in Layout 7-366

3D Layout

interface ports and sources 9-49

push excitations 9-65, 9-71

push excitations to 9-65

3D line, drawing 7-222

3D models

changing background 13-2

clip planes 13-29

curved surface 13-28

fitting 13-20

hiding objects 13-21

lighting 13-33

modifying view 13-1

orientation 13-17

panning 13-8

rendering solids 13-24

rendering wireframes 13-24

rotating 13-18

spinning 13-18

user interface options 13-2

zooming 13-9, 14-64

3D Overlay, Animating 14-540, 14-540, 14-543, 14-547

3D Placement 6-115

3D Report, Spinning 14-475

3D structures

create 7-366

managing 11-56

ungroup 7-366

3D view

overlaying far fields 14-478

overlaying near fields 14-480

overlaying surface currents 14-476

A

aborting an analysis 10-694

abstract class

IProgressMonitor 14-583

optional functions 14-552

accessing

include files 5-223

library files 5-226

netlist files 5-221

ACT extensions 2-59, 2-176

activating layers 7-190

adaptive analysis

and frequency sweeps 10-448

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bypassing 10-3, 10-5, 10-13

setting maximum number of passes 10-13

adaptive passes, completed and remaining 14-10

adaptive solution

method for computing 18-4

additional .NET assemblies 14-565

admittance matrix

method of calculation 18-12

admittance matrix, plotting parameters 14-512

aligning objects in Layout 7-170

AMI

properties 5-142

AMI analysis 5-162

outputs 5-163

running 5-163

AMI analysis, HFSS 3D Layout 10-433

analysis

aborting 10-694

AMI 5-162

distributed 10-486

parametric (large scale DSO) 10-504

remote 10-478

running AMI 5-163

analysis configurations

in the registry 3-66

Analysis Setup, enable/disable 10-477

analyze all 15-3

ANF file format 5-1

exporting to 5-98

ANFV2 design data, importing 5-2

ANFV4 design data, importing 5-3

ANFV4, Translations of Padstacks to 11-93

Animating a 3D Overlay 14-540, 14-540, 14-543, 14-547

animations 14-269

anisotropic tensors 8-10

AnsTranslator

import nets 5-44

Running from within Designer 5-85

Running on the Command Line 5-42

Ansys Electronics Desktop

batchoptions 2-148, 2-165, 2-173

files 3-1

getting started with 2-1

launching 2-5

menu bar 2-82

overview 2-11

projects 3-1

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ribbon interface 2-12, 2-177, 3-13
running from a command line 2-148, 2-165, 2-173
running from Windows remote terminal 2-176
status bar 2-137
system requirements 2-5
user interface 2-11
window displays 2-16
Ansys Fast solver, operation of 18-5
Ansys Product Improvement Program 2-8
Ansys technical support 1-40
Ansys Workbench
feedback iterator 16-46
integration with 16-1
material data transfer 16-35
antenna properties
overview 14-210
apertures, drawing on negative signal planes 7-215
API, user-defined Python scripts 14-574
archive
restoring 3-89
saving project as 3-86
arcs
drawing 7-221
editing in Layout 7-186
assemblies, additional .NET 14-565
assign reference port 7-256
Auto Layer Placement 7-256
auto-aligning objects 7-172
AutoCAD, DXF import 5-14
automated CPM 5-218
auto-save 3-81

B

backdrills
adding 11-107
settings 11-109
background color 13-2
basis function 18-2
batchoptions 2-148, 2-165, 2-173
command line examples 3-62
desktop settings 2-162
directories and library paths 2-161
examples 2-158
file format 3-62
in the registry 3-66
override registry entry 2-164
temp directory 2-161
bathtub curves, reports 14-597

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 3-5

board outlines 5-1

bondwires

drawing 7-28

Bondwires 7-29

bottom-associated layer 7-200

bricks, dielectric 7-259

browser, capacitor/inductor library 7-303

buffer element, import 5-117, 5-117, 5-119, 5-121, 5-122, 5-124, 5-127

buffer import tab 5-127

bus symbol pins, editing 5-202

C

Cadence APD Links 5-40, 5-41

cadence files, importing 5-16

calculations, UDO for dimensions reduction 14-550

Calibration Wizard 5-199, 5-199

capacitor/inductor library browser 7-303

changing

the 2D Design Point 14-547

characteristic impedance

and renormalized S-parameters 18-10

characteristic impedance, HFSS 3D Layout 18-9

characteristic impedance, plotting 14-512

charts, 2D and Circuit Smith 14-443

chip model analyzer (CMA) 14-612

Choose a Layout Technology window 7-98

circle, drawing 7-219

circuit

elements 9-41

inserting Ensemble 1-3

circuit eye diagram reports, modifying 14-518

circuit models, importing 5-98

circuit netlist

include files 5-223

library files 5-226

Circuit PAM4 Eye Plot 14-530

circuit ports

, creating 9-46

in layout 7-258

circular polarization

generating on a patch with a single feed 11-102

class, constants 14-582

classes, python scripts 14-552, 14-553, 14-553, 14-554, 14-555, 14-555

clean stop 10-694, 15-5

cleaning up solutions 14-31

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

clear reference port 7-256
clip plane 13-29
clip planes 7-53, 13-32
clipboard
  options 3-30
closing
  Project Manager 2-23
  projects 3-101
clustering
  vias 7-248
coax ports 9-28
coherent gain 14-469
color scheme
  changing 3-9
colors of layers 7-97
combine sweeps 14-21
command properties
  modifying 2-32
command-line syntax 2-148, 2-165, 2-173
commands
  hide objects 13-2
  pan 13-8
  redo 3-95
  rotate view 13-18
  show objects 13-2
  spin view 13-18
  undo 3-95
  zoom 13-9, 14-64
complex
  loads 7-235, 7-237
component
  command 7-2, 7-262
  create 7-263, 7-298
  creating verilog 5-208
  creating, editing 11-11
  creation sequence 11-9
  edit IBIS properties 5-133
  editor 11-10
  encapsulation, layout based 7-266
  libraries 11-1
  new member 7-305, 7-305, 7-307
  parameters, reserved 11-25
Component CosimDefinition property 11-26
component creation 6-115, 14-2
component libraries 6-9
components
  creating, editing 11-11, 11-12
  editing 11-12
  editing definitions 6-20
  editing properties 6-18
  symbol footprint 7-255
  updating 3-94

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

viewing properties of 6-14
window 2-42, 7-83
conductors, estimating loss 11-96
configuration files
example uses 3-52
user options 3-54
using to set options 3-43
using UpdateRegistry command 3-47
connecting pins 7-87
connections, drawing 7-254
constants
list of 3-116
constants class 14-582
contour plots
creating 14-447
smith 14-448
control file
ODB++ 5-45
usage with renaming 5-49
control file for ODB++, via Excel 5-52
convergence criteria
maximum number of passes 10-13
convergence data
maximum delta Energy results 14-11
maximum delta S results 14-10
number of passes completed 14-10
plotting 14-11
convert planes to traces 7-54
convert trace to plane 7-54
converting, between common microwave parameters 11-106
coordinate systems 7-368, 7-373
create 7-369
managing 11-56
copy and paste
report data 14-506
trace data 14-506
copy command, for report and trace definitions 14-505
cosimulation
field solver designs 3-173
co-simulation
options, LNA 10-427
coupling ports 9-12
CP patches, estimating 11-102
CPM model support 5-217
CPU time
viewing for solution tasks 14-3
vs. real time 14-3
CPW
creating 9-12

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

slotline ports 9-40

Create Components with 3D Placement 6-115, 14-2

create pin groups 7-39

creating

components 11-9, 11-11

crcuit port 9-46

edge port 9-2

mesh plot 17-60

new footprint 11-50

new padstack 11-82

new symbol 11-30

report from file 14-457

cross-layer plates, drawing 7-235

cross-probing

selected elements 7-217

curves

settings 3-25

tooltip 3-30

customize

user tools 2-145

cutout subdesign 7-72

D

data format, UserDefinedDocument 14-588

data markers, adding to trace 14-490

data tables, creating 14-438, 14-442

data types, in Python scripts 14-552, 14-582

dataset expressions 3-171

datasets

adding 3-142

cloning 3-150

editing 3-148

exporting 3-152

importing 3-146

overview 3-142

plot properties 3-153

removing 3-152

using SheetScan 3-155

dB

converting from 11-106

power calculation 11-106

voltage calculation 11-106

DC IR 7-373

DC IR in HFSS 3D Layout 10-243, 10-265, 10-272, 10-274, 10-280, 10-289

DC thickness, assigning 8-1

debug logging 2-179

Debye's model 18-15

de-embedded

theory and S-Matrices 18-12

de-embedding, S-matrices 9-61

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default layers 7-197

defaults

basic field quantities 14-267

define footprints, using scripts 11-61

define traces, using range functions 14-493

defining

footprint handles 11-60

definition archives

exporting 11-3

importing 11-6

importing and exporting 11-3

definition layer, initial padstack 11-92

definition libraries 11-1

definitions, updating on the library 11-3

deleting

layers 7-192

reports 14-486

delta between markers, in reports 14-472

design

placement 7-84

point, changing 14-547

design area 2-134

designs

copying and pasting 3-85

deleting 3-93

design notes 3-98

nominal 12-1

read-only 3-93

setting up 1-1

updating components 3-94

viewing details in the project tree 2-25

DesignXplorer 12-163

desktop configuration 3-5

targeted 3-5

desktop-engine connection 3-19

diagrams, stacked eye 14-527

dielectric layers

and signal layers 7-213

avoiding air pockets 7-213

bricks 7-259

modeling 7-259

dielectric merging 7-197

dielectric simplification 7-197

dielectrics, estimating loss 11-96

differential pairs 9-65

reports 14-456

digital voltage levels, IBIS file import 5-149

dimensions reduction, UDO calculations 14-550

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direct Import, Cadence APD Links 5-39
directional drawing, layout editor 7-218
directive gain 14-515
directive gain, antenna properties 14-210
directories
options 3-15
system, user and personal 11-2
discard below value 14-491
discontinuities and modes 18-13
display
Layout Editor 7-9
distributed analysis
command line options 10-542
configuration 10-487
optimal settings 10-502
overview 10-486
two-level distribution 10-503
distributed machine 1-27
distributed machine configurations 1-27
Djordjevic-Sarkar causal dielectric 18-21
documentation file
inserting 3-98
documents
generator interfaces 14-590
scripting interface 14-587
user-defined inputs 14-567
draw menu, symbol 11-32
Draw>Connection command 7-254
Draw>N-Port command 7-249
Draw>Primitive commands
Arc 7-221
Circle 7-219
Image 7-220
Line 7-230
Polygon 7-232
Rectangle 7-219
Draw>Void commands 7-35, 7-233
drawing region
fitting in viewing area 7-214
grid settings 7-214
model overview 7-214
overview 7-214
shadow settings 7-215
size of 7-214
snap settings 7-214
drop-down menus
Layout Editor View Menu 2-96
Schematic Editor View Menu 2-95

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DSO
large scale 10-504
command line syntax 10-513
configuration 10-522
monitoring 10-521
outputs 10-517
post-processing 10-519
prerequisites 10-506
troubleshooting 10-533
user interface 10-507
duplicate (objects) command 7-177, 7-179
duplicating objects 7-178, 7-179
duplicating objects, in XY 7-177
DXF export, AutoCad 5-93
DXF import, AutoCad 5-14
dynamic link
push excitation to 9-70
dynamic probes 14-550

E

ECAD Xplorer 1-3, 5-30, 7-122
EDB cells, programmatic 6-115
edge object
methods 11-77
properties 11-76
edge objects 11-76
edge port, creating 9-2
edge port, removing 9-12
edge ports, connecting to N-ports 7-254
edges, selecting 7-172
Edit Components window 11-12
general tab 11-12
miscellaneous tab 11-14
Solver on Demand tab 11-18
terminals tab 11-16
Edit drop-down menus
Layout Editor 2-92
Schematic Editor 2-91
Edit Layers window 7-90, 7-187, 7-350, 7-352
color, visibility, patterns 7-97
elevations 7-201
HFSS 3D Layout 1-3
layout technology 7-98
Edit Libraries window 11-138
Edit Material window 11-124
Edit Padstack Definition window 11-82, 11-88
edit pin groups 7-43
Edit Port Excitations window 9-56
edit solution 10-477
edit symbol pins 11-42
Edit>Mirror X command 7-174
Edit>Mirror Y command 7-174

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

editing
components 11-12
materials 11-124
Editing Pin Properties 11-41
editor
component 11-10
layout options 7-10, 7-17
library 11-137
material 11-124
models 11-123
padstack 11-82
script 11-49
symbol 11-29
windows 2-105
ElementPars object 11-71
elements
cross-probing selected 7-217
w-elements support 5-163
elevation
and non-trace layers 7-213
Edit Layers window 7-201
lower 7-202
upper 7-202
EM subdesign plotting 14-604
encapsulation, layout based 7-266
encrypted
libraries 11-129, 11-132
passwords 11-132
Environment Variables
PinToPin utility 14-638
equivalent circuit export options 14-24
errors
disk space warning 3-16
errors layer 7-197
errors,logging 2-179
Estimate module 11-95
Microwave Calculator commands 11-106
Rectangular Patch commands 11-102
estimating
characteristic impedance 11-96
CP element parameters 11-103
effective relative permittivity of traces 11-96
guided wavelength of traces 11-96
line loss 11-98
matches between impedances 11-99
nearly square CP element 11-102
Printed Antenna commands 11-102
printed antennas 11-102
quarter-wave transformers 11-99
rectangular patches 11-102
transmission line quantities 11-96
tuning stubs 11-99

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

width of traces 11-96

etch factor

calculator 6-27

controls 7-96

setting 7-203

Excel generated ODB++ control file 5-52

excitation tree 7-373

excitations

across multiple layers 9-46

creating coupled ports 9-12

creating pad ports 9-14

reference nearest ground 9-53

reference surface 9-52

two terminal 9-52

varying the port 9-63

Expand object command 7-178

export

DXF 5-93

NC drill and router 5-96

export stride 6-6

exporting 5-90

ANF layout 5-98

data table files 5-90

datasets 3-152

definition archives 11-3, 11-3

equivalent circuit data files 14-24

graphics files from 3D 5-97

hierarchical components 11-140

layout data 5-91

LNA results 5-218

materials 8-37

netlist data 5-221

netlist files 5-229

network design data 5-218

options files 3-103

schematic data 5-98

to SSS File 5-220

W-element data 14-27

Exporting Hierarchical Components 11-140

expressions

dataset 3-171

intrinsic functions 3-168

operators 3-167

overview 3-166

piecewise linear functions 3-170

expressions, using variables 7-177

extension implementation, UDO 14-551

external tools, customize 2-145

eye diagrams 14-518

PAM4 14-530

plot 14-449

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

plots 14-448
stacked 14-527
eye measurements
2D and circuit 14-521
range function 14-527
eye plot 14-451
eye source
HFSS 3D Layout 10-440

F

far fields
overlaying on 3D view 14-478
plotting quantities 14-514
fast calculation-update 12-144
fast frequency sweeps
modifying matrix data 14-538
Fast sweeps, modifying matrix data 14-12
FFT (Fast Fourier Transform) 14-469
field overlays
plotting 14-262
field quantities
defaults 14-267
phase angle 14-309
field solver designs, co-simulation 3-173
fields calculator
complex vector electric field 14-309
general commands 14-306
input commands 14-296
output commands 14-321
overview 14-287
registers 14-293
scalar commands 14-310
stack commands 14-290
vector commands 14-315
Fields Calculator Cookbook 14-313
file formats
.amat 11-124
.gds 5-5
.m files 14-17
.sm2 5-5
.sm2, EDB Cadence, ANF, AutoCad, GDSII, ODB++, IPC2581a, XFL 5-5
.sNp 14-19
.spc 14-24
data table 14-19
Ensemble ver. 6+ 14-17
HFSS ver. 6+ 14-17
Libra 14-19
Neutral Model Format 14-17
nmf 14-20

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Touchstone 14-19
file section, paths for 5-201
files
about 3-1
importing 5-9, 5-10
finite element mesh, MPIE 18-1
fitting objects 13-20
flipping 2D objects 7-174
fonts
scalable and nonscalable 7-76
stroke, TrueType 7-75
footprint
creating 11-50
editing existing 11-51
editor 11-50
load into project 11-51
scripts to define 11-61
stackup 6-96
Footprint Edit Menu 11-51
footprint handles, defining 11-60
Footprint View Menu 11-53
footprint, handles 11-60
frequency
and propagating modes 18-14
dependent materials 18-15
frequency points
choosing for full-wave SPICE 10-458
Frequency Sweep
disable/enable 10-459
range for full-wave SPICE 10-458
frequency sweeps
adding to designs 10-448
and adaptive analysis 10-448
and bypassing adaptive analysis 10-3, 10-5, 10-13
Interpolating sweep overview 18-6
settings for 10-448
frequency-dependent materials
data points 8-23
debye model 8-18
defining 8-12
djordjevic-sarkar model 8-22
multipole debye model 8-19
piecewise linear 8-16
frequency-dependent materials, and lossy dielectrics 18-15
full-wave SPICE
choosing frequencies 10-458
overview 18-14
set up 7-51
sweep ranges 7-44

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

functions
defining 3-166
intrinsic 3-168
list of 3-166
piecewise linear 3-170
reports 14-506
selecting for a quantity 14-507

G

Gap(coax) ports 9-28
GDSII format files, importing 1-3, 5-18, 5-30, 7-122, 10-468
geom object
properties 11-73
geometry
check in layout 7-395
Gerber
format files, importing 5-8
global
coordinate system 7-368
layout options 7-10
grid
settings 7-214
grid engine (GE) 10-569
grids
settings 3-26
Groiss surface roughness 7-210
ground plane, model signal 6-97
grouping
vias 7-248
guided wavelength, estimating 11-96

H

handles
selecting 7-170
using 7-169
harmonic loss 10-460
healing geometric objects 7-179
help 2-35
HFSS
distributed machine, HFSS 3D Layout 1-27
etch factor 6-29
mesh errors 17-90
mesh simplification 17-84
Model Extents 10-45
near field solution setup 10-198, 10-224, 10-324
pi solution setup 10-55, 10-70, 10-295
pin-based ports 9-13
port types 9-1
ports for imported Package 9-13
ports for PCB Structures 9-13
ports for transmission lines 9-2
ports, validating connectivity 9-54
solution setup 10-1, 17-33

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
distributed machine 1-27
eye sources and probes 10-440
HFSS distributed machine 1-27
HFSS Mesh Errors 17-90
HFSS Mesh Simplification 17-84
HFSS Model Extents 10-45
inserting designs 1-3
modes 18-13
optimization 10-448
port setup 9-1
properties 7-44
QuickEye and VerifyEye 10-440
QuickEye Nexxim options 10-442
QuickEye outputs 10-444
QuickEye setup 10-441
run Quick Eye analysis 10-444
sanitize layout 10-465
SIwave distributed machine 1-28
SIwave solution setup 10-151, 10-152, 10-155, 10-158, 10-193
solution log file 18-14
technical notes 18-1
VerifEye Nexxim options 10-446
HFSS in Designer 6-1
hfss pi 10-55, 10-70
HFSS solutions 10-1, 10-55, 10-70, 10-89, 10-198, 10-224, 10-294, 10-324, 17-33
hide selection 13-21
hiding
layers 7-97
windows 2-16
hierarchical components, exporting 11-140
hierarchical design 6-1, 7-262, 7-366, 7-368, 7-373
high performance computing
configuring Ansys Electronics for 10-546
diagnostics 10-688
grid engine (GE) 10-569
integration 10-544
load sharing facility (LSF) 10-580
Microsoft Windows HPC 10-547
overview 10-478
portable batch system (PBS) 10-590
remote simulation management (RSM) 10-665
schedulers 10-545
terminology 10-544
third party schedulers 10-599
holes, drawing 7-235
horizontal lumped ports 9-34, 9-55

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

horizontal wave ports 9-31

HPC

setting options 3-32

HSPICE, exporting to 14-24

Huray surface roughness 7-211

I

IBIS

file import, technical notes 5-143

library file import 5-115

node thresholds 5-151

specification 5-143

IBIS component properties 5-133, 5-133, 5-135, 5-137, 5-138, 5-138, 5-139, 5-141, 5-141

IBIS file import

digital voltage levels 5-149

input node thresholds 5-149

output node thresholds 5-150

Package RLC Parasitics 5-144

power aware 5-145

power nodes 5-143

technical notes 5-151

transition setting times 5-151

TYP parameters 5-155

waveform properties 5-152

IBIS file import, technical notes 5-143, 5-144, 5-145, 5-149, 5-149, 5-150, 5-151, 5-152, 5-155

IBIS I/O Wizard

power/ground setup 5-160

Signal Setup 5-159

source/analysis setup 5-162

window 5-158

IBIS Properties window

AMI Properties 5-142

IBIS single component, import 5-117, 5-117, 5-119, 5-121, 5-122, 5-124, 5-127

Icepak 7-373

IDO Extension, optional functions 14-552

image, drawing 7-220

impedance

characteristic 18-9

estimating on transmission lines 11-96

matching 11-99

matrix, calculation 18-11

renormalizing S-matrices 9-59

renormalizing S-parameters 18-10

impedance matrix, plotting parameters 14-512

impedance, for N-port data 7-253

import

file (symbol) 11-40

IBIS library file 5-115

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

single component, no package, all pins 5-121

single component, no package, selected pins 5-122

single component, with pacakage, all pins 5-117

single component, with package, selected pins 5-119

statements, 2D and circuit 14-574

import 3d cad 6-2, 6-8

import IBIS single component 5-117, 5-117, 5-119, 5-121, 5-122, 5-124, 5-127

import nets 5-44

import, pin or buffer element 5-117, 5-117, 5-119, 5-121, 5-122, 5-124, 5-127

imported solutions

data, post-processing 5-242

noise data tab 5-234

N-port data tab 5-231

options tab 5-178, 5-236

importing

2D model files to Layout Editor 5-5

ANF file format 5-1

ANFV2 design data 5-2

ANFV4 design data 5-3

cadence files 5-16

circuit models 5-98

datasets 3-146

definition archives 11-3, 11-6

DXF and DWG 5-14

EDB 5-6

GDSII format filies 1-3, 5-18, 5-30, 7-122, 10-468

Gerber format filies 5-8

IBIS .ebd files 5-156

idf files 5-9

IPC2581 revisions a and b 5-13

Layout design data 5-1

model 5-101

multi-file state-space model 5-181

netlist data 5-221

network design data 5-183

network solution data 5-230

N-port data files 7-252

N-ports 7-251

ODB++ 5-11

PSPICE library procedure 5-106

solution data for postprocessing 5-230

SPICE library 5-104

stackup 7-260

state-space models 5-175

table arrays 5-37

tabular data 5-238

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

XFL files 5-18
x-parameter component 5-165
include files
circuit netlists 5-223
independent stackups 6-98
independent variables, x-parameters 5-173
infinity visualization 14-466
initial
padstack definition layer 11-92
input
interfaces, UUD 14-584
node thresholds 5-149
insert
circuits, HFSS 3D Layout 1-3
inserting circuit
HFSS 3D Layout 1-3
interactive scheduler jobs
design types 10-541
DSO configuration 10-537
options 10-535
overview 10-535
interface
document generator 14-590
ports and sources 9-49
UUD input 14-584
interpolating
sweep 18-6
sweeps, modifying matrix data 14-538
Interpolating sweeps, modifying matrix data 14-12
interpolation method, for calculating N-port data 7-249
intersecting objects 7-177
Intersection Checks for 3D Components 7-279
IProgressMonitor, abstract class 14-583
IUDDPluginExtension, abstract class 14-575

J

jitter reports 14-601
jobs
aborting 10-694
advanced submission options 10-636
exporting 10-641
importing 10-641
monitoring 10-656
multi-step submission 10-641
scripting 10-654
submitting 10-631
submitting using command line 10-598, 10-640
Windows to Linux 10-647
joining objects 7-175

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
- custom 2-143
- desktop 2-139
- general 2-139
- layout editor 2-140
- report window 2-141

L

Laminate stackup 6-31

large scale DSO 10-504
- command line syntax 10-513
- configuration 10-522
- monitoring 10-521
- outputs 10-517
- post-processing 10-519
- prerequisites 10-506
- troubleshooting 10-533
- user interface 10-507

layers
- activating 7-190
- adding 7-192
- default 7-197
- defining 6-30
- drawing on 7-200
- elevation 7-201, 7-201, 7-201
- freezing 7-190
- hiding from view 7-97
- mapping files 5-65
- matching circuit 7-200
- merge-layers window 7-82
- modifying 6-30
- order 7-201, 7-201, 7-201
- removing 7-192
- setting associations 7-200
- setting colors 7-97
- setting patterns 7-97
- setting thickness 7-202
- setting visibility 7-97
- system and plotter 7-75
- visibility 2-65, 7-188
- working with 7-187, 7-350, 7-352

layout
- circuit ports 7-258
- context menu 7-76
- draw menu 7-25
- dropdown menu 7-58
- edit menu 7-21
- host object 11-61
- import and export 5-1
- links 5-40
- sanitize 10-465
- stackup 6-29
- technology, choosing 7-98, 7-102

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

technology, Edit Layers window 7-98

view menu 7-24

view, zoom and pan 7-215

layout component

command 7-307

create 7-317, 7-327

layout components

editing definitions 7-337

Layout Editor 7-1

default display 7-9

directional drawing 7-218

display and menus 7-9

importing GDSII format files 1-3, 5-18, 5-30, 7-122, 10-468

importing Gerber format files 5-8

importing ODB++ 5-11

importing XFL files 5-18

negative objects 7-173

nets drop-down menu 7-66

options 7-10, 7-11, 7-13, 7-14, 7-15, 7-16, 7-17, 7-20

selecting overlappig objects 7-172

selection modes 7-168

vias and pin padstacks 11-91

view menu 2-96

voids 7-217

windows 7-82

layout host methods 11-62, 11-62, 11-63, 11-66, 11-67, 11-69

Layout menus

context 7-76

draw 7-25

edit 7-21

view 7-24

layout options

global 7-10

local 7-20

layout window 2-135

LayoutHost properties 11-61

legacy, line style attributes 7-74

legends

settings 3-27

legends in reports 14-468

length of transmission line

adding to or subtracting from a port 9-61

libraries

changing locations 11-2

component 11-1

definition 11-1

encrypted 11-129

importing SPICE libraries 5-103

personal 8-39

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

script 14-565

search path precedence 11-9

system 8-39

user 8-39

library

browser, capacitor/inductor 7-303

components, RF vendor 11-128

editor 11-137

files, in circuit netlists 5-226

precedence 11-2

structures and usage precedence 11-2

vendor components 11-128

Verilog 5-204

Library Editor 11-137

lighting 13-33

lightweight geometry 6-1, 6-2, 6-6, 6-8

limit lines in plots 14-472

line

ending types 7-75

style attributes, legacy 7-74

styles 7-74

width 7-231

line loss, estimating 11-96

line ports 9-10, 9-10, 9-11, 9-11

line, drawing 7-230

linear constraints 12-154

linear network analysis

solution setup 10-426

Linux remote spawn 3-39

LNA

cosimulation options 10-427

setup netlist 10-427

solution setup 10-426

load sharing facility (LSF) 10-580

load types, for vias 7-235, 7-237

loading

a footprint 11-51

local layout options 7-20

log of solution tasks 14-3

logging errors 2-179

loop tree discretization, overview 18-3

lower elevation, assigning to layers 7-202

M

MAFET Consortium 14-20

Manage Files 11-8

Map Infinity Mode 14-466

markers

delta between markers 14-472

settings 3-28

table settings 3-28

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 settings 3-28
matching layers 7-200
materials
adding new 8-7
anisotropic tensors 8-10
assigning 8-1, 8-33
assigning properties 8-10
assigning to layers 7-197
copying 8-36
creating 11-128
deleting 8-36
Edit Material window 11-124
editing 11-127
editor 11-124
exporting 8-37
ferrite 8-7
filtering 8-37
frequency-dependent 8-12
functional properties 8-33
libraries 8-39
linear 8-7
modifying attributes 8-33
properties as expressions 8-32
removing 8-36
searching 8-3
sorting 8-37
thermal modifiers 8-25
validating 8-36
variable properties 8-12
viewing attributes 8-33
viewing definitions 2-24
MATLAB optimization 3-8
matrices
admittance
computing from S-parameters 18-12
method of calculation 18-12
impedance
calculation 18-11
relationship to S-parameters 18-10, 18-10
matrix data
display format 14-16, 14-539
exporting 14-18
modifying frequencies 14-12, 14-538
renaming 14-21, 14-540
reordering 14-22
viewing 14-538
maximum delta Energy, viewing results 14-11
maximum delta S, viewing results 14-10
maximum number of passes, setting 10-13
Maxwell Spice files, exporting to 14-24
mcad component 6-1, 6-2, 6-8
measures layer 7-197

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

memory, used during solution 14-3
menu bar 2-82
menus
customizing 2-125
design-specific 2-105
draw menu (layout editor) 2-107
edit menu 2-86
edit menu (3D Layout) 2-86
edit menu (netlist editor) 2-89
edit menu (report window) 2-90
file menu 2-84
help menu 2-119
Layout Draw 7-25, 7-76
Layout Edit 7-21
Layout Editor 7-9
layout pulldown 7-58
layout view 7-24
project menu 2-104
shortcut menus 2-122
tools menu 2-108
adding tools 2-111, 2-115
view menu 2-93
view menu (3D Layout) 2-99
view menu (basic) 2-93
window menu 2-117
merge polygons 7-38
merge-layers window 7-82
merging, dielectric 7-197
mesh
3D placement 17-58
errors 17-90
HFSS 17-1
length-based 17-61
seeding 17-61, 17-61, 17-63
simplification 17-84
size vs. accuracy 18-3
skin-depth 17-63
mesh generation
process 17-54
mesh refinement
length based 17-61
meshing
generating mesh without solve 17-49
Phi Plus 17-79
settings 17-26, 17-33
statistics 14-30
message manager 2-26
action messages 2-28
automatically expanding 2-27
clearing messages 2-28
hiding 2-28
showing new messages 2-27

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

messages
UDOs (user-defined outputs) 14-558
metallized signal layers, and signal layers 7-213
method of moments
basis function 18-2
compressing matrix 18-5
re-ordering of matrix 18-3
testing function 18-2
Microsoft Windows HPC
commands 2-176
integration with 10-547
microstrip line, estimating purely resistive input impedance 11-99
Microwave Calculator 11-106
Minerva remote storage environment 4-1
mirroring objects 7-174
mismatch loss, converting from 11-106
model
selecting to import 5-101
signal, ground plane 6-97
support, automated CPM 5-218
support, of w-elements 5-163
support, state-space 5-174
modeling, dielectric 7-259
Models Editor, using 11-123
modes
HFSS 3D Layout 18-13
non-propagating (evanescent) 18-14
modify solution 10-477
monitor job 10-656
MPI Vendor, selecting 3-41
MPIE method
finite element mesh 18-1
implementation 18-4
loop tree discretization 18-3
method of moments 18-2
representation of current 18-2
size of mesh vs accuracy 18-3
multi-file
state-space model, editing 5-182
state-space model, importing 5-181
multi-step job submission 10-641
Multizone stackup 6-31
creating 6-32

N

named probes and properties 14-549
NC drill and router, export 5-96
near field 10-198, 10-224, 10-324
near fields, overlaying on 3D view 14-480
negative objects, Layout Editor 7-173

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

negative signal layer 6-29

negative signal planes, drawing apertures on 7-215

neither-associated layer 7-200

Net Importation in AnsTranslator 5-44

netlist

data importing and exporting 5-221

include files 5-223

setup LNA 10-427

syntax 11-26

Nets

dropdown menu, Layout > Nets 7-66

nets window 2-43, 2-67

network data explorer 14-638

cell filtering 14-658

exporting data 14-643

loading data 14-642

menu commands 14-654

modifying the display 14-669

overview 14-639

smoothing 14-657

network design data

exporting 5-218

importing 5-183

new report window 14-434

Nexxim, optimzation 10-448

node thresholds, IBIS file imports 5-151

noise

data tab, imported solutions 5-234

noise control reports 14-601

nonscalable fonts 7-76

n-port

and data files 7-252

creating mode with no file 5-188

creating model 5-184

creating parametric 5-190

editing bus symbol pins 5-202

editing model 5-185

editing multi-file model 5-187

guidelines for inserting 7-251

imported solutions 5-231

multi-file model 5-185

phase rotation 7-254

placing 5-185

two-port models 9-41

n-port model (parametric), editing 5-192, 5-192, 5-193, 5-196, 5-196, 5-199

n-port model, editing 5-176, 5-177, 5-178, 5-181, 5-236

N-ports

changing the reference

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

impedance 7-253
connecting to edge ports 7-254
inserting in projects 7-249
manually entering data 7-253
number of passes
setting maximum 10-13

O

object, ElementPars 11-71
objects
aligning, auto-aligning 7-170
bondwires 7-28
creating from intersections 7-176
drawing 7-214, 7-219
duplicating 7-177, 7-178, 7-179
editing 7-174
fitting 13-20
flipping or mirroring 7-174
healing 7-179
hiding 13-21
Layout Editor 7-168
modifying attributes 2-32
opening 7-219
rotating 7-174
showing and hiding 13-2
solid 13-24
wireframe 13-24
ODB++
control file 5-45
control file, Excel 5-52
control file, part mapping 5-87
exporter issues 5-58
importing to Layout 5-11
Translator 5-85
opening
objects 7-219
operators
list of valid 3-167
optimetrics
linear constraints 12-154
link to DesignXplorer 12-163
optimization analysis 12-16
overview 12-1
parametric analysis 12-2
sensitivity analysis 12-121
statistical analysis 12-134
viewing results 12-157
optimization
choosing an optimizer 12-16
overview 12-16
setting up analysis 12-47
optimization analysis
Fast Calculation-Update 12-144

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

optimization, subcircuit 10-448
optional functions, IDO Extension 14-552
options
3D user interface 13-2
animation 3-16, 3-24
auto-save 3-81
axis 3-25
beta 3-5
clipboard 3-30
component 3-21
curve 3-25
digital 3-30
exporting as file 3-103
general 3-3
component libraries 3-21
default units 3-19
desktop configuration 3-5
desktop performance 3-16
directories 3-15
miscellaneous 3-8
project 3-7
remote analysis 3-19
user interface 3-9
global layout 7-10
grid 3-26
HPC and analysis 3-32
layout editor 7-10, 7-17
legend 3-27
local layout 7-20
marker table 3-28
notes 3-26
pre/post-processing 3-16
report headers 3-26
report markers 3-28, 3-28
report table 3-31
report update 3-16
Report2D 3-24
reporter 3-22
fields reporter 3-24
report setup 3-23
reports 3-30
RSM 3-19
setting via configuration files 3-43, 3-47
significant digits 3-23
stacked 3-30
options tab, imported solutions 5-178, 5-236
options, co-simulation LNA 10-427
options, report 2D 14-459
options, report setup 14-458
options, Symbol Editor 11-31
output node thresholds, IBIS file support 5-150

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 14-252
deleting 14-254, 14-538
specifying 14-536
outputs, AMI analysis 5-163
outputs, user-defined Python script API 14-551
overlapping objects, selecting 7-172
Overlapping stackup 6-96
overlaying far fields, 3D view 14-478
overlaying near fields, 3D view 14-480
overlaying surface currents, 3D view 14-476

P

Package RLC Parasitics, IBIS file import 5-144
pad behavior, vias and pins 11-90
pad ports, creating 9-14
padstack
creating new 11-82
definitions
converting primitives 7-242
Edit Padstack Definition window 11-82, 11-88
Padstack Definition Translations 11-93
padstack editor 11-82
padstact
initial 11-92
pan and zoom, Layout view 7-215
parameter
reserved 11-22
reserved component 11-25
statistics display 11-23
parameter defaults
adding from components 3-132
converting to variables 3-131
defining 3-123
deleting 3-123
editing 3-123
overriding 3-123
parametric n-port model, editing 5-192, 5-192, 5-193, 5-196, 5-196, 5-199
parametric n-port, creating 5-190
password manager 2-147
passwords, setting up 11-132
paths for file selection 5-201
pattern matching, List window 7-61
patterns, of layers 7-97
personal library 8-39
personal library directories 11-2
personallib 11-2

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

phase
rotation, and N-ports 7-254
Phi Plus Mesher 17-79
pin
connectivity window 7-87
creating pin-based ports 9-13
editing bus-symbol 5-202
element, import 5-124
groups 7-39
groups, create 7-39
groups, edit 7-43
groups, rename 7-44
locations, editing 11-42
padstacks and vias
editing 11-91
properties, editing 11-41
pin and net operations 11-93
pin element, import 5-117, 5-117, 5-119, 5-121, 5-122, 5-127
pin groups 7-39
pins
converting primitives 7-242
PinToPin utility
command line 14-616
environment variables 14-638
Linux 14-617
using 14-612
window 14-618
XML 14-629
place
design window 7-84
IBIS buffer component, schematic 5-128
plane, convert to trace 7-54
Plot-On-Schematic 14-454
plots
2D and Circuit 3D Polar 14-446
2D and Circuit 3D Rectangular 14-445
2D and Circuit Plot-On-Schematic 14-454
2D and Circuit Polar 14-441
2D and Circuit Rectangular 14-439
2D and Circuit Rectangular Stacked 14-444
2D and Circuit Rectangular Contour Plot 14-447
2D and Circuit Reports With Differential Pairs 14-456
2D and Circuit Smith Contour Plot 14-448
2D polar 14-111
2D rectangular 14-93
2D rectangular stacked 14-95
3D polar 14-113
3D rectangular 14-97

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

adding limit lines 14-472
Circuit Eye Diagram 14-448
Circuit Stacked Eye Diagram Plot 14-449
Circuit Statistical Eye Plot 14-451
data tables 14-125
limit lines in 14-159
radiation patterns 14-127
rectangular contour 14-104
sine space 14-107
smith charts 14-122
smith contour charts 14-123
time domain 14-129
y markers 14-144
plots on schematic, creating 14-438
plotting
characteristic port impedances 14-513
EM subdesign 14-604
VSWR 14-513
points object 11-71
methods 11-72
properties 11-71
polar plots
2D and Circuit 3D 14-446
information displayed 14-475
polar plots, 2D and Circuit 14-441
polygon
curving an edge 7-232
port
create, footprint editor 11-56
excitations, edit 9-56
parametric n-port 5-190
remove, footprint editor 11-56
port setup
HFSS 3D Layout 9-1
portable batch system (PBS) 10-590
ports 9-13
coax 9-28
coupling 9-12
CPW and Slotline 9-40
creating coupled ports 9-12
creating pad ports 9-14
Exposing in Hierarchy 9-55
horizontal lumped 9-34
horizontal wave 9-31
port-instance 9-15
post-processing many ports 14-517
sources, in 3D Layout 9-49
wave 9-29
position locking 7-173
post processing
importing solution data for 5-230

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 options 14-1
viewing convergence data 14-7
viewing matrix data 14-12
viewing profile data 14-3
post-processing
imported solution data 5-242
set up 9-55
transient in layout 14-606
viewing matrix data 14-538
post-processing, handling many ports 14-517
power
aware, IBIS file import 5-145
power distribution 7-373
power electronics 10-460
power nodes, IBIS file import 5-143
power/ground setup
IBIS I/O Wizard 5-160
precedence prioritizing 7-47
precedence, library usage 11-2
preview images, choosing 3-83
primary sweep
modifying the variable 14-506
specifying for 3D polar plots 14-438
primitives
converting 7-242
printing 3-100
a Layout 7-396
pro, premium, enterprise licensing 3-5
probes
HFSS 3D Layout 10-440
probes and properties 14-549
probes, dynamic 14-550
probing, cross-probing 7-217
profile information, viewing 14-3
programmatic EDB cells 6-115
progress window 2-29
project manager window 2-20, 2-21
project manager, tree organization 2-22
Project manger, closing 2-23
project tree
automatically expanding 2-23
overview 2-23
viewing design details 2-25
visualization options 3-9
project, loading footprint into 11-51
projection view 13-35
projects
about 3-1
archiving 3-86
closing 3-101
copying and pasting 3-85

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

creating new 3-77
deleting 3-93
design notes 3-98
example projects 1-30, 3-101
legacy projects 3-75
opening 3-71
renaming 3-86
saving 3-79
validating 3-95
working with 3-1
propagation constant
and de-embedding 18-12
propagation constant, plotting 14-512
Properties
defining for components 11-22
pin, editing 11-41
report backgrounds 14-466
properties
via object 11-77
properties dialog box 2-37
general and symbol tabs 2-40
opening 2-38
parameter values tab 2-38
property displays tab 2-40
properties window 2-30
automatically opening 2-32
modifying command properties 2-32
modifying object attributes 2-32
opening 2-31
showing and hiding 2-32
properties, edit IBIS component 5-133
properties, editing trace 14-488
property display setup, symbol 11-47
Property Window for Schematic and Layout Editors 2-33
propoerties, for UDO 14-552
PSpice files, exporting to 14-24
PSPICE library
import procedure 5-106
push escitations, to sub-designs 9-70
push excitations 9-65
Thevenin Equivalent 9-71
push excitations, to 3D Layout ports 9-65
push excitations, to dyamic links 9-70
Python script API, user-defined definitions 14-574
Python script API, user-defined outputs 14-551
Python scripts, .NET classes and interfaces 14-556
python scripts, application specific classes 14-552, 14-553, 14-553, 14-554, 14-555, 14-555
Python scripts, data types 14-552, 14-582

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Q

Q3D 10-93

quantities

plotting far field 14-514

plotting field 14-513

plotting S-parameter 14-512

reports 14-506

quarter-wave transformer, estimating dimensions 11-99

Quick Eye analysis

run in HFSS 3D 10-444

QuickEye and VerifyEye

HFSS 3D Layout 10-440

QuickEye outputs

HFSS 3D Layout 10-444

QuickEye setup

HFSS 3D Layout 10-441

HFSS 3D Nexxim options 10-442

QuickEye, plotting bathtub curves 14-597

R

radiation efficiency 14-515

RAM, used during solution 14-3

range function parameters, eye measurements 14-527

range function, setting 14-516

range functions 14-211

range functions, define traces 14-493

rats layer 7-197

read-only designs 3-93

rectangle, drawing 7-219

rectangular patch, estimating resonating length 11-102

rectangular plot, 2D and Circuit 14-439, 14-445

rectangular plots 14-438

rectangular stacked plots, creating 14-444

Rectangular Contour Plot 14-447

redo command 3-95

reduction, of dimensions via UDO calculations 14-550

reference impedance, for N-port data 7-253

reference nearest ground 9-53

reference ports

assign 7-256

clear 7-256

Reference to Nearest Ground 9-53

references, x-parameter 5-174

reflection

of higher order modes 18-13

reflection coefficient, converting from 11-106

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Regions Schematic 10-186, 10-188, 10-190

registry entry

overriding 2-164

regular expressions 5-86

relative coordinate system 7-368

operations affecting 7-368

release

obtaining information about 2-121

remote analysis

overview 10-478

troubleshooting 10-481

remote simulation management (RSM) 10-665

remote spawn (Linux) 3-39

remove material from object 7-35, 7-233

removing

edge port 9-12

removing layers 7-192

rename pin groups 7-44

renaming, matrix data 14-21, 14-540

renormalization of S-matrices 18-10

renormalizing S-matrices 9-59

reordering matrix data 14-22

replicating objects 7-177, 7-179

report 2D options 14-459

report data, copy and paste 14-506

report options, setting 2D and circuit 14-458

report properties, discard below value 14-491

report setup options 14-458

report types 14-433

report window 2-134

report, adding a trace 14-487

Report2D Options

Axis Tab 14-459

Curve Tab 14-459

Digital Tab 14-464

General Tab 14-464

Grid Tab 14-460

Header Tab 14-460

Legend Tab 14-461

Marker Tab 14-461

Marker Table Tab 14-462

Table Tab 14-465

X/Y Markers Tab 14-462

Report2DOptionsNoteTab 14-460

reporter window types 14-469

reports

2D and Circuit Creating a Report from a File 14-457

adding traces 14-486

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

animated 14-236

background properties 14-466

bathtub curves 14-597

context 14-38

creating 14-33

creating 2D and circuit 14-432

custom templates 14-83, 14-89

data tables 14-125

display types 14-438

file formats 14-88

finding delta between markers 14-472

handling many ports 14-517

jitter and noise controls 14-601

modifying 14-42

modifying data in 14-465

modifying properties 14-68, 14-78

perform FFT 14-217

perform TDR 14-222

plotting far-field quantities 14-514

plotting field quantities 14-513

selecting a function 14-507

Smith charts 14-438

sweeping variables 14-506

types of 14-90

user defined documents (UDDs) 14-398

user defined outputs (UDOs) 14-356

user-defined outputs 14-548

variables and quantities and functions 14-506

zooming and fitting 14-64

Reports With Differential Pairs 14-456

reports, deleting 14-486

reports, generating for 2D and circuit 14-432

reports, modifying 2D and circuit 14-465

reports, modifying circuit eye diagram 14-518

reports, technical notes for 2D and circuit 14-596

reserved component parameters 11-25

results documents, managing in the Project Manager 14-570

results window 2-134

return loss, converting from 11-106

Reverse Line Command 7-232

Revert to Initial Mesh 17-59

RF vendor, library components 11-128

rise time, minimum 10-458

rotating 2D objects 7-174

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rotating the phase, for N-port data 7-254

roughness column, stackup tab 7-209

run commands 2-148, 2-165, 2-173

running, AMI analysis 5-163

S

sanitize layout, HFSS 3D Layout 10-465

save options 3-5

scalable and nonscalable fonts 7-76

schedulers

- Ansys RSM 10-665
- GE 10-569
- integration using IronPython 10-621
- LSF 10-580
- Microsoft Windows HPC 10-547
- overview 10-545
- PBS 10-590
- proxy interfaces 10-610
- selecting 10-629
- terminology 10-544
- testing custom integration 10-618
- testing integration 10-615
- third party 10-599

schematic

- creating on plots 14-438

Schematic Editor

- View menu 2-95

script editor 11-49

script libraries, using 14-565

script libraries, UUD 14-573

script organization, user-defined outputs 14-564

scripts, define footprints 11-61

secondary sweep

- modifying the variable 14-506
- specifying for 3D polar plots 14-438

Select Coordinate System window 7-369

selecting

- edges 7-172
- handles 7-170

selecting layers 7-190

selection modes, layout editor 7-168

sensitivity analysis 12-121

- choosing variables 3-139

setting up designs 1-1

setting up projects 1-1

setup netlist, LNA 10-427

shadow settings, choosing 7-214

SheetScan 3-155

- checking monotonicity 3-163

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 3-161

exporting data 3-165

importing characteristic data 3-163

loading datasheets 3-158

shortcut menus 2-122

3D modeler 2-123

project manager 2-124

short-open-load-thru method, Touchstone Calibration Wizard 5-201

signal layers

and dielectric layers 7-213

and metallized signal layers 7-213

signals, minimum rise time 10-458

significant digits

setting 3-23

simplification, dielectric 7-197

simulations

aborting 2-29

changing solution priority 15-5

monitoring 15-4

monitoring queued 15-3

re-solving after modifying design 15-6

re-solving after thermal feedback 15-6

running 15-1

running multiple 15-3

stopping 2-29, 15-5

single frequency solutions, specifying 10-3, 10-5, 10-13

Single Mesh with 3D Placement 17-58

SIwave

solution setup in HFSS3D 10-151, 10-152, 10-155, 10-158, 10-193

SIwave distributed machine

HFSS 3D Layout 1-28

SIwave export 7-373

SIwave Region Based Simulations 10-177

SIwave Regions 10-186, 10-188, 10-190

SIwave SYZ in HFSS 3D Layout 10-192

SIwave Workflow Wizard 10-152, 10-155, 10-158, 10-193

SIwave-PSI 10-295

S-matrices

de-embedding 9-61

overview 18-9

renormalizing 9-59

Smith charts, creating 14-438, 14-443

Smith Contour Plot 14-448

snap settings, choosing 7-214

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solution
method of moments 18-2
solution data
deleting 14-32
viewing 14-3
viewing convergence data 14-7
solution data, importing for postprocessing 5-230
solution frequency
and Fast sweeps 10-10
solution log file
HFSS 3D Layout 18-14
solution process
tetrahedra used during 14-3
viewing memory used during 14-3
viewing profile data 14-3
viewing tasks performed 14-3
solution settings
adding frequency sweeps 10-448
maximum number of passes 10-13
solution frequency 10-10
solution setup 10-1, 10-477
adding to project 10-1
solution setup, linear network analysis 10-426
solutions
aborting 15-5
changing priority 15-5
electrothermal flow 10-460
monitoring 15-4
re-solving after modifying design 15-6
re-solving after thermal feedback 15-6
solving 15-1
SolveIndependent 17-58
Solver On Demand Tab 11-18
Solver Options window 7-89
solving
inside an object 8-1
Solving Inside 8-1
Solving on Surface 8-1
source promotion 7-373
S-parameters
and frequency 18-9
de-embedded 18-12
de-embedding 18-12
overview 18-9
plotting quantities 14-512
renormalized 18-10
SPB License Requirements 5-41
spectral 14-469

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SPICE 10-155
full-wave SPICE overview 18-14
SPICE library
import procedure 5-104
SPICE pin nmes, modifying 5-108
SPICE RLC 10-158
Spinning a 3D Report 14-475
square patch
estimating corners-truncated CP element 11-102
generating orthogonal modes 11-102
Stacked Eye Diagram Plot 14-449
stacked eye diagrams 14-527
stacked plots, creating 14-444
stackup
adding a layer 7-192
adding a zone 6-41
adding layers 7-192
configuring 6-30
copying a zone 6-36
creating a Multizone 6-32
creating a zone 6-36
deleting layers 7-192
Laminate 6-31
laminate 7-201
layer elevation 7-201, 7-201, 7-201
layer order 7-201, 7-201, 7-201
modifying 6-30
Multizone 6-31
multizone 7-201
Overlapping 6-96
overlapping 7-201
removing a zone 6-44, 6-63, 6-77, 6-81, 6-86
stackup definitions 6-27
Stackup tab
Lower Elevation column 7-202
Material Properties column 7-197
Thickness column 7-202
Upper Elevation column 7-202
stackup tab
Roughness column 7-209
Stackup Wizard, Differential Impedance 6-108, 6-110, 7-55
Stackup Wizard, HFSS 3D 6-99
Stackup Wizard, Impedance 6-107
Stackup Wizard, Insertion Loss, Return Loss, Time Domain Crosstalk 6-112
Stackup Wizard, Trace Width, Trace Pitch 6-102
stackup, importing 7-260
stackups, independent 6-27
state-space model support 5-174

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

state-space model, editing 5-176, 5-176, 5-177, 5-178, 5-181, 5-236

state-space model, editing multi-file 5-182

state-space model, import to create 5-181

state-space models, importing 5-175

statistical analysis 12-134

choosing variables 3-140

Statistical Eye Plot 14-451

status bar 2-137

Status Bar 2-138

step and repeat, duplicating objects 7-177, 7-179

stopping criteria

maximum number of passes 10-13

Stride 6-1, 6-2, 6-6, 6-8

stroke font 7-75

subcircuit optimization 10-448

sub-designs, push excitations to 9-70

submit job 10-631

substate stackup 6-27

subtracting objects 7-176

surface currents, overlaying 14-476

surface force density 16-62

surface roughness 7-96, 7-209

surface roughness model, Groiss 7-210

surface roughness model, Huray 7-211

surface roughness, assigning to layers 7-209

surfaces

curved 13-28

sweep types 7-50, 10-454

sweep variables in reports, modifying values 14-506

sweeps

orphaned 3-135

variable 3-135

Symbol draw menu 11-32

symbol editor 11-29

Symbol Editor options 11-31

symbol footprint components 7-255

symbol grid, setup window 11-48

symbol layer, Auto Layer Placement 7-255

Symbol Menu 11-38

symbol pins, edting n-port bus 5-202

symbol property display setup 11-47

symbol, creating new 11-30

symbol, editing existing 11-31

symbol, import file 11-40

symbol, verilog 5-205

symbols layer 7-197

syslib 11-2

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

system and plotter layers 7-75

system directories 11-2

system library 8-39

system resources

changing solution priority 10-693

requirements 2-5

SYZ in HFSS 3D Layout 10-162

syz-parameters, importing 5-183

T

tables, creating data 14-442

tabular ASCII data files 5-90

tabular data, importing 5-238

TDR plots, HFSS 3D Layout 10-1, 10-1

TDR plots, HFSS 3D layout 10-426

technical notes, 2D and circuit reports 14-596

technical notes, HFSS 3D 18-1

technical notes, IBIS file import 5-143, 5-143, 5-144, 5-145, 5-149, 5-149, 5-150, 5-151, 5-151, 5-152, 5-155

technical notes, x-parameter 5-168

technical support 1-40

technology files, HFSS 3D Layout 1-4

technology, layout, choosing 7-98

temporary directory

configuration file format 3-60

setting 3-58

using UpdateRegistry command 3-61

terminal assignments 9-28

Terminals Tab, Edit Components window 11-16

testing function 18-2

tetrahedra

used during solution 14-3

text styles 7-75

thermal feedback 15-6

thermal simulation 7-373

Thevenin Equivalent

push excitations with 9-71

thickness of layers 7-202

thin film properties 7-50

Thru-reflect-line method

Touchstone Calibration Wizard 5-200

time

real vs. CPU 14-3

viewing for solution tasks 14-3

time steps per rise time 10-458

top-associated layer 7-200

Touchstone Calibration

two-line method 5-199

Touchstone Calibration Wizard

Thru-reflect-line method 5-200

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Touchstone Calibration Wizard, short-open-load-thru method 5-201

trace characteristics, 2D and circuit 14-492

trace data, copy and paste 14-506

trace properties, editing 2D and circuit 14-488

trace, adding data markers to 14-490

trace, adding to a report 14-487

trace, convert to plane 7-54, 7-54

traces

- adding characteristics to 14-151
- adding data markers 14-141
- adding to reports 14-486
- copy and paste definitions 14-505
- define using range functions 14-493
- display properties 14-489
- editing properties 14-137
- overview 14-136
- removing 14-157, 14-493
- y markers 14-144

traces based computation, UDO calculations 14-550

transient analysis, HFSS 3D Layout 10-429

Transient Excitation 10-261

transient in layout, post-processing 14-606

transition setting times, IBIS file import 5-151

Translations, of Padstacks 11-93

Transmission Line Ports 9-10, 9-10, 9-11, 9-11

transmission lines

- estimating 11-96
- estimating effective relative permittivity 11-96
- estimating impedance 11-96
- estimating line loss 11-98
- estimating the guided wavelength 11-96
- estimating width 11-97
- via HFSS ports 9-2, 9-2

tree organization, project manager 2-22

triangles, calculating current inside 18-2

TrueType font 7-75

TSV Layers 7-194

tuning 3-138, 12-148

tuning stub, estimating location and dimensions 11-99

two-level distribution

- guidelines 10-503

two-line method

- Touchstone Calibration Wizard 5-199

two-port n-port models 9-41

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

two-way coupling 7-373

TYP parameters, IBIS file import 5-155

U

UDDs, viewing with HTML browser 14-572

UDO calculations, dimensions reduction 14-550

UDO calculations, traces based computation 14-550

UDO extension implementation 14-551

UDO, properties 14-552

undo command 3-95

uniting objects 7-175

units of measure

setting default 3-19

upper elevation, assigning to layers 7-202

user defined documents (UDDs) 14-398

user defined outputs (UDOs) 14-356

user directories 11-2

user library 8-39

user tools, customize 2-145

user-defined definitions, Python script API 14-574

user-defined document, window inputs 14-567

user-defined Documents (UDDs) 14-566, 14-566

user-defined Outputs 14-548

user-defined outputs, messaging methods 14-558

user-defined outputs, named probes and properties 14-549

user-defined outputs, Python script API 14-551

user-defined outputs, script organization 14-564

user-defined, document scripting interface 14-587

UserDefinedDocument data format 14-588

userlib 11-2

Using Definition Libraries 11-1

Using the Component Editor 11-10

Using the Library Editor 11-137

Using the Material Editor 11-124

Using the Padstack Editor 11-82

Using the Script Editor 11-49

Using the Symbol Editor 11-29

UUD, document creation and display 14-569

UUD, input interfaces 14-584

UUD, script libraries 14-573

UUDextemsion class 14-575

UUDInputParams class 14-582

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

V

validation check 3-95
variables
adding from components 3-132
constants 3-116
copying and pasting list 3-141
design-level 3-117
arrays 3-125
converting to parameter defaults 3-131
defining 3-118
editing 3-118
overriding 3-118
parameter defaults 3-123
fixed vs. non-fixed 3-135
in sensitivity analysis 3-139
in statistical analysis 3-140
intrinsic 3-115
modifying value of a fixed 12-153
optimizing 3-136
output 14-252, 14-536
overview 3-104
project-level
adding 3-105
arrays 3-110
deleting 3-105
editing 3-105
reports 14-506
sweeping 14-166
tuning 3-138, 12-148
used in expressions 7-177
varying the port excitation 9-63
Vendor Library Components, components, vendor library 11-128
VerifEye setup
HFSS 3D Nexxim options 10-446
VerifyEye and QuickEye
HFSS 3D Layout 10-440
VerifyEye, plotting bathtub curves 14-597
verilog component, creating 5-208
Verilog features supported 5-213
Verilog library support 5-204
verilog symbol, creating 5-205
vertex, adding to an edge 7-186
via layers 7-213
via object 11-77
methods 11-78
properties 11-77
vias
and pin padstacks
edit in layout 11-91
backdrilling 11-107
clustering 7-248

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

converting primitives 7-242
drawing 2.5D vias 7-235, 7-237
grouping 7-248
load types for 2.5D 7-241
overview 7-235, 7-237
stubs 11-107
vias and pins, pad behavior 11-90
view navigation options 2-141
view options
rotate 13-18
visibility, setting for layers 7-97
voids, creating 7-35, 7-233
voids, in layout 7-217
voltage standing wave ratio
plotting 14-512
voltage standing wave ratio, converting from 11-106
VSWR, converting from 11-106

W

wave port 9-29
waveform properties, IBIS file import 5-152
W-element data, exporting 14-27
w-element, model support 5-163
window displays
ACT extensions 2-59
components 2-42, 7-83
layout window 2-135
message manager 2-26
nets 2-43, 2-67
progress window 2-29
project manager 2-20
properties window 2-30
report window 2-134
results window 2-134
status bar 2-137
windows
auto-hiding 2-16
docking 2-16
moving and resizing 2-16
showing and hiding 2-16
windows, Layout Editor 7-82
wirebond layer 7-197
wizard, calibration 5-199
working coordinate system 7-368
working with objects
Layout Editor 7-168

X

XFL files, importing to Layout Editor 5-18
x-parameter component, importing 5-165
X-parameter model import 5-165
x-parameter, references 5-174

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

x-parameter, technical notes 5-168

x-parameters, independent variables 5-173

Y

Y-parameters, plotting 14-512

Z

zone

add 6-41

copying 6-36

creating 6-36

deleting 6-63, 6-77, 6-81, 6-86

removing 6-44

zoom and pan, Layout view 7-215

zooming 13-9, 14-64

Z-parameters, plotting 14-512

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.